

User Manual



ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
<http://www.ansys.com>
(T) 724-746-3304
(F) 724-514-9494

ANSYS ICEM CFD 12.1
November 2009

ANSYS, Inc. is certified to ISO 9001:2008.
--

Copyright and Trademark Information

© 2009 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. is certified to ISO 9001:2008.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the [legal information](#) in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

Introduction to ANSYS ICEM CFD	1
Overall Process	1
Opening/Creating a Project	2
Creating/Manipulating the Geometry	3
Creating the Mesh	3
Checking/Editing the Mesh	4
Generating the Input for the Solver	5
Postprocessing	5
The ANSYS ICEM CFD GUI	5
GUI Components	6
Main Menu	6
Utilities	7
Function Tabs	7
The Display Control Tree	8
The Message Window	9
The Histogram Window	9
The Data Entry Zone (DEZ)	10
Using the Help System	11
CAD Repair	15
Close Holes	15
Remove Holes	16
Fill, Trim and Blend in Stitch/Match Edges	17
Match in Stitch/Match Edges	18
Tetra Meshing	21
Introduction	21
Tetra Mesh Generation	21
Input to Tetra	22
Tetra Generation Steps	22
Repairing the Geometry	23
Geometry Details Required	23
Sizes on Surfaces and Curves	24
Meshing Inside Small Angles or in Small Gaps Between Objects	24
Desired Mesh Region	24
The Octree Mesh Method	25
Important Features in Tetra	28
Curvature/Proximity Based Refinement	28
Tetrahedral Mesh Smoother	28
Tetrahedral Mesh Coarsener	28
Triangular Surface Mesh Smoother	29
Triangular Surface Mesh Coarsener	29
Triangular Surface Editing Tools	29
Check Mesh	29
Smooth Mesh Globally	29
Quality Metrics	31
Advanced Options for Smoothing Mesh	31
Prism Mesh	33
Prism Mesh Process	34
Prism Mesh Preparation	34
Smoothing Tetra/Prism Mesh	35
Hexa	37

Introduction	37
Features of Hexa	37
Mesh Generation with Hexa	38
The Hexa Database	38
Intelligent Geometry in Hexa	39
Unstructured and Multi-block Structured Meshes	39
Unstructured Mesh Output	39
Multi-Block Structured Mesh Output	39
Blocking Strategy	39
Hexa Block Types	40
Split	41
Merge	41
Using the Automatic O-grid	41
Most Important Features of Hexa	42
Automatic O-grid generation	43
Important Features of an O-grid	43
Edge Meshing Parameters	43
Smoothing Techniques	44
Refinement and Coarsening	45
Refinement	45
Coarsening	45
Replay Functionality	45
Generating a Replay File	45
Advantage of the Replay Function	45
Using Variables in the Replay Script	45
Periodicity	46
Applying the Periodic Relationship	46
Pre-Mesh Quality	46
Determining the Location of Elements	46
Determinant	46
Angle	47
Volume	47
Warpage	47
Properties	49
Create Material Property	49
Save Material	49
Open Material	49
Define Table	49
Define Elements	49
Constraints	51
Create Constraint / Displacement	51
Define Contact	51
Define Single Surface Contact	51
Define Initial Velocity	51
Define Planar Rigid wall	51
Loads	53
Force	57
Pressure	57
Temperature	57
Solve Options	59
Setup Solver Parameters	59
Setup Analysis Type	59

Setup Sub-Case	59
Write/View Input file	59
Submit Solver Run	59
Post Process Results	59
FEA Solver Support	59
Cart3D	61
Workbench Integration	63

Introduction to ANSYS ICEM CFD

ANSYS ICEM CFD provides advanced geometry acquisition, mesh generation, mesh optimization, and post-processing tools to meet the requirement for integrated mesh generation and post processing tools for today's sophisticated analyses.

Maintaining a close relationship with the geometry during mesh generation and post-processing, ANSYS ICEM CFD is used especially in engineering applications such as computational fluid dynamics and structural analysis.

ANSYS ICEM CFD's mesh generation tools offer the capability to parametrically create meshes from geometry in numerous formats:

- Multiblock structured
- Unstructured hexahedral
- Unstructured tetrahedral
- Cartesian with H-grid refinement
- Hybrid meshes comprising hexahedral, tetrahedral, pyramidal and/or prismatic elements
- Quadrilateral and triangular surface meshes

ANSYS ICEM CFD provides a direct link between geometry and analysis. In ANSYS ICEM CFD, geometry can be input from just about any format, whether from a commercial CAD design package, 3rd party universal database, scan data or point data. Beginning with a robust geometry module which supports the creation and modification of surfaces, curves and points, ANSYS ICEM CFD's open geometry database offers the flexibility to combine geometric information in various formats for mesh generation. The resulting structured or unstructured meshes, topology, inter-domain connectivity and boundary conditions are then stored in a database where they can easily be translated to input files formatted for a particular solver.

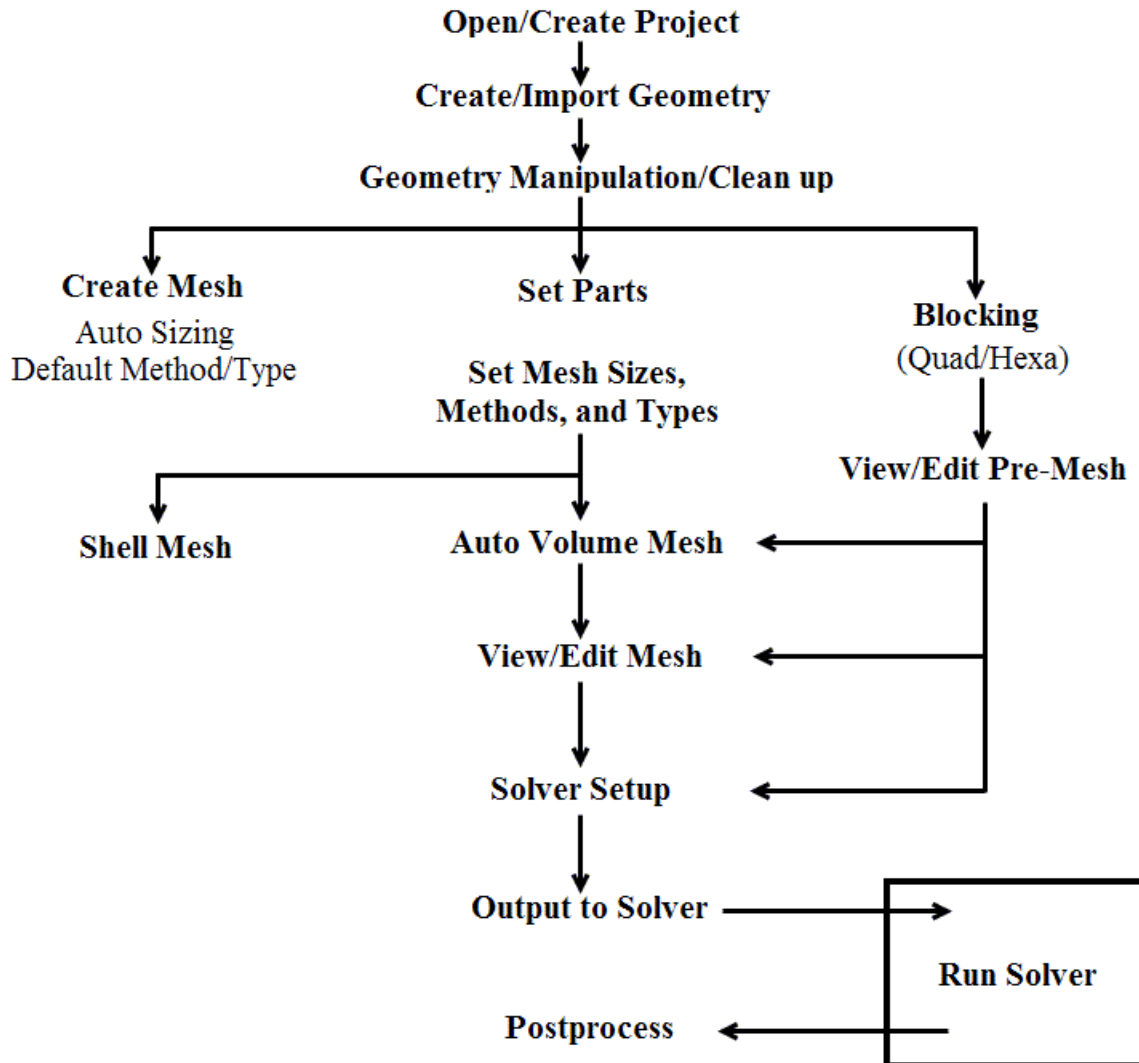
[Overall Process](#)

[The ANSYS ICEM CFD GUI](#)

Overall Process

The generic working process involves the following:

1. Open/Create a project.
2. Create/Manipulate the geometry.
3. Create the mesh.
4. Check/Edit the mesh.
5. Generate the input for the solver.
6. Postprocess the results.

Figure: The Overall Process

Opening/Creating a Project

All the files required for a particular analysis are contained within a **Project**. You can either open an existing project or create a new project. The **Project** directory typically contains one or more of the following file types:

Tetin (*.tin)

contains geometry entities, material points, part association, and global and entity mesh sizes.

Project Settings (*.prj)

contains the project settings.

Domain (*.uns)

contains the unstructured mesh.

Blocking (*.blk)

contains the blocking topology.

Boundary Conditions (*.fbc)

contains boundary conditions.

Attributes (*.atr)

contains attributes, local parameters, and element types.

Parameters (*.par)

contains model parameters and element types.

Journal (*.jrf)

contains a record of operations performed (echo file).

Replay (*.rpl)

contains the replay script.

Creating/Manipulating the Geometry

ANSYS ICEM CFD includes a wide range of tools for creating new and/or manipulating existing geometry. You can either alter complex geometry or create simple geometry without having to go back to the original CAD. This can be done for CAD (NURBS surfaces) and triangulated surface data. The ANSYS ICEM CFD Direct CAD Interfaces provide the bridge between parametric geometry creation tools available in CAD systems and the computational mesh generation, postprocessing and mesh optimization tools available in ANSYS ICEM CFD, allowing users to operate in their native CAD systems. ANSYS ICEM CFD currently supports Direct CAD Interfaces for CATIA, I-deas, Pro/E, and Unigraphics.

The ANSYS ICEM CFD environment can combine CAD surface geometry and triangulated surface data into a single geometry database (tetin file) using the geometry interfaces. All geometry entities, including surfaces, curves and points are tagged or associated to a grouping called a part. With this part association, you can enable or disable all entities within the parts, visualize them with a different color, assign mesh sizes on all entities within the part and apply different boundary conditions by part.

Although most of the meshing modules within ANSYS ICEM CFD allow minor gaps and holes in the geometry, in some cases it is necessary to find and close large gaps and holes without returning to the original CAD software. ANSYS ICEM CFD provides tools for such operations on either CAD or triangulated surfaces. Finally, curves and points can be automatically created to capture certain key features in the geometry. These curves and points will act as constraints for the mesher, forcing nodes and edges of the elements to lie along them, and thus capturing the feature.

Creating the Mesh

The meshing modules available include the following:

Tetra

The ANSYS ICEM CFD Tetra mesher takes full advantage of object-oriented unstructured meshing technology. With no tedious up-front triangular surface meshing required to provide well-balanced initial meshes, ANSYS ICEM CFD Tetra works directly from the CAD surfaces and fills the volume with tetrahedral elements using the Octree approach. A powerful smoothing algorithm provides the element quality. Options are available to automatically refine and coarsen the mesh both on geometry and within the volume. A Delaunay algorithm is also included to create tetras from an existing surface mesh and also to give a smoother transition in the volume element size.

Hexa

The ANSYS ICEM CFD Hexa mesher is a semi-automated meshing module which allows rapid generation of multi-block structured or unstructured hexahedral volume meshes. ANSYS ICEM CFD Hexa represents a new approach to grid generation where the operations most often performed by experts are automated and made available at the touch of a button. Blocks can be built and interactively adjusted to the underlying CAD geometry. This blocking can be used as a template for other similar geometries for full para-

metric capabilities. Complex topologies, such as internal or external O-grids can also be generated automatically.

Prism

ANSYS ICEM CFD Prism generates hybrid tetrahedral grids consisting of layers of prism elements near the boundary surfaces and tetrahedral elements in the interior for better modeling of near-wall physics of the flow field. Compared to pure tetrahedral grids, this results in smaller analysis models, better convergence of the solution and better analysis results.

Hybrid Meshes

The following types of hybrid meshes can be created:

- Tetra and Hexa meshes can be united (merged) at a common interface in which a layer of pyramids is automatically created at a common interface to make the two mesh types conformal. These meshes are suitable for models where it is preferred to have a “structured” hexa mesh in one part and is easier to create an “unstructured” tetra mesh in another more complex part.
- Hexa-Core meshes can be generated where the majority of the volume is filled with a Cartesian array of hexahedral elements essentially replacing the tetras. This is connected to the remainder of a prism/tetra hybrid by automatic creation of pyramids. Hexa-Core allows for reduction in number of elements for quicker solver run time and better convergence.

Shell Meshing

ANSYS ICEM CFD provides a method for rapid generation of surface meshes (quad and tri), both 3D and 2D. Mesh types can be **All Tri**, **Quad w/one Tri**, **Quad Dominant**, or **All Quad**. The following methods are available:

- **Mapped based shell meshing (Autoblock):** Internally uses a series of 2D blocks, resulting in a mesh better lined up with geometry curvature.
- **Patch based shell meshing (Patch Dependent):** Uses a series of “loops” which are automatically defined by the boundaries of surfaces and/or a series of curves. This method gives the best quad dominant quality and capturing of surface details.
- **Patch independent shell meshing (Patch Independent):** Uses the Octree method. This is the best and most robust method for unclean geometry.
- **Shrinkwrap:** Used for quick generation of mesh. As it is used as the preview of the mesh, hard features are not captured.

Checking/Editing the Mesh

The mesh editing tools in ANSYS ICEM CFD allow you to diagnose and fix problems in the mesh. You can also improve the mesh quality. A number of manual and automatic tools are available for operations such as conversion of element types, refining or coarsening the mesh, smoothing the mesh, etc.

The process typically involves the following:

1. Check the mesh for problems such as holes, gaps, overlapping elements using the diagnostic checks available. Fix the problems using the appropriate automatic or manual repair methods.
2. Check the elements for bad quality and use smoothing to improve the mesh quality.
3. If the mesh quality is poor, it may be appropriate to fix the geometry instead or recreate the mesh using more appropriate size parameters or a different meshing method.

Generating the Input for the Solver

ANSYS ICEM CFD includes output interfaces to various flow and structural solvers, producing appropriately formatted files that contain complete mesh and boundary condition information. After selecting the solver, you can modify the solver parameters and write the necessary input files. The list of supported solvers is available at the [ANSYS ICEM CFD website](#).

Postprocessing

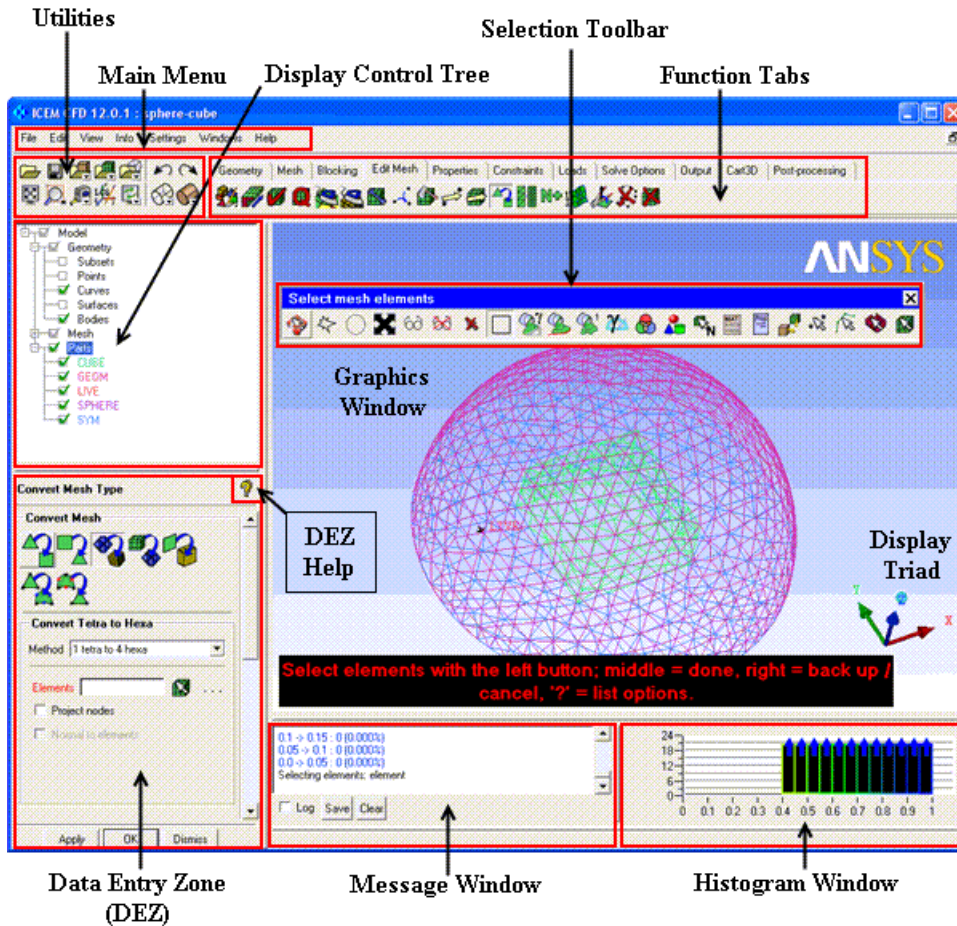
ANSYS ICEM CFD allows you to view the solution results from different solvers. The results file (from various CFD and structural formats) must first be loaded to make the postprocessing menu active. You can visualize results using cut planes, iso-surfaces, imported external surfaces, streams, animations, calculate integrals, and plot XY files.

The ANSYS ICEM CFD GUI

The ANSYS ICEM CFD GUI offers a complete environment to create and edit computational grids. The main menu is in the top left corner. Below it are utility icons for more commonly used functions such as **Save** and **Open**, as well as measure tools and view controls such as zoom extents. Along the top right of the window are function tabs. The function tabs are laid out from left to right in the order of a typical meshing process. Clicking on a tab brings its action icons to the fore front. Clicking on any of these icons will activate the associated **Data Entry Zone (DEZ)**. The snapshot shows the **Convert Mesh Type** DEZ which also has a selection toolbar associated with it. The histogram is displayed in the lower-right corner. Simultaneously, the text histogram is displayed in the message window. The message window provides feedback and information for most commands and also serves as a text entry point. The upper left corner of the screen contains the **Display** tree, which you can use to modify the display of entities, modify properties and create subsets.

Note

The default GUI style shown in [Figure : ANSYS ICEM CFD GUI Components \(p. 6\)](#) is the **Workbench** style. For more information about the **GUI Style** options, refer to the [Product-Selection](#) settings.

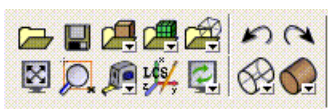
Figure: ANSYS ICEM CFD GUI Components

GUI Components

The various GUI components are described in the following sections:

- Main Menu
- Utilities
- Function Tabs
- The Display Control Tree
- The Message Window
- The Histogram Window
- The Data Entry Zone (DEZ)

Main Menu

Figure: The Main Menu

The **Main Menu** provides access to the following pull-down menus:

File Menu

contains options for creating new or opening existing projects, loading and saving files, importing and exporting geometry, and initializing scripting.

Edit Menu

contains **Undo/Redo** options, the option to open a shell window, and various internal mesh/geometry conversion commands.

View Menu

contains various options for the standard views, view controls, and annotations.

Info Menu

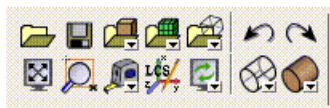
allows you to get various information regarding geometry, mesh and individual entities.

Settings Menu

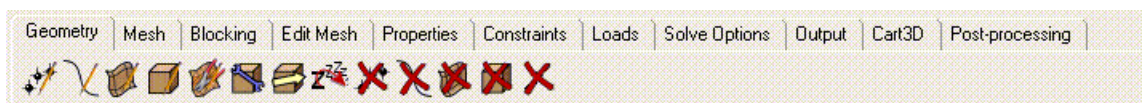
contains default settings for performance, graphics, and other settings most likely to be used more than 90% of the time by a specific user.

Help Menu

contains links to Help Topics, tutorials, the User's Guide and version information.

Utilities**Figure: Utilities**

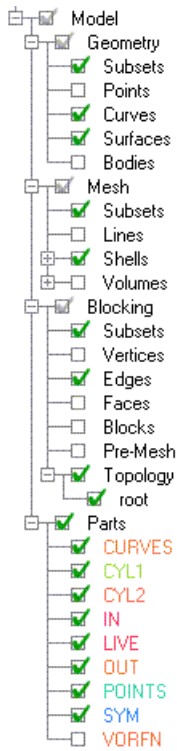
The **Utilities** are icon representations of some of the most commonly used functions in the **Main Menu** including opening/closing a project, undo/redo, and display options. They also include measurement and setup of local coordinate systems.

Function Tabs**Figure: Function Tabs**

The **Function Tabs** allow you to access the main functionality for the entire grid generation process. The function tabs include: **Geometry**, **Mesh**, **Blocking**, **Edit Mesh**, **Properties**, **Constraints**, **Loads**, **Solve Options**, **Output**, **Cart3D**, and **Post-processing**.

The Display Control Tree

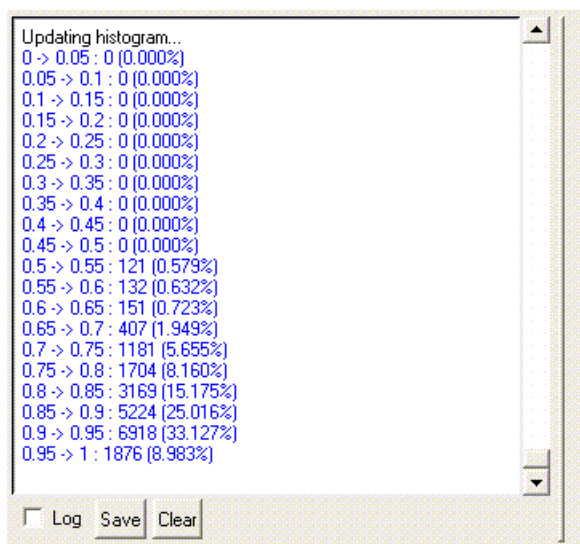
Figure: The Display Control Tree



The **Display Control** tree, also referred to as the **Display** tree, along the upper left side of the screen, allows control of the display by part, geometric entity, element type and user-defined subsets. The tree is organized by categories. Each category can be enabled or disabled by selecting the check box. If the check mark is faded, some of the sub-categories are enabled and some disabled. Each category can be expanded by selecting the "+" symbol to reveal the sub-categories. Select "-" to collapse the tree. Since some functions are performed only on the entities shown, the tree is an important feature to use when isolating the particular entities to be modified. Clicking on a particular category or type using the right-mouse button will reveal several display and modification options.

The Message Window

Figure: The Message Window



The **Message Window** contains all the messages that ANSYS ICEM CFD writes out to keep the user informed of internal processes. The **Message Window** displays the communication between the GUI and the geometry and meshing functions. You can review the information to be informed of the status of operations. Any requested information, such as measure distance, surface area, etc. will be reported in the message window. Also, internal commands can also be typed and invoked within the message window.

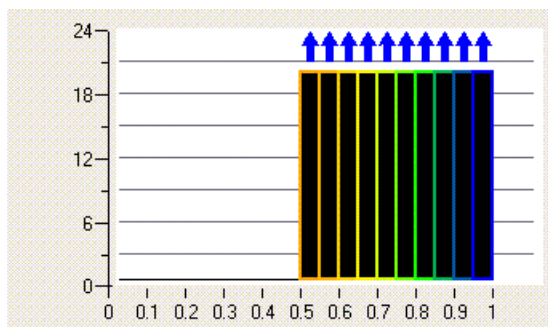
The **Save** command will write all message window contents to a file. This file will be written to wherever ANSYS ICEM CFD was launched. The **Log** check box allows only user specified messages to be saved to a file.

Note

The **Log** file is unique from the file created with the **Save** button. This file will be written to the starting directory, and it interactively updates as more messages are recorded. Once the check box is disabled, you can continue to add to the file by enabling **Log** and accepting the same file name (which is the default). It will then continue to append this file.

The Histogram Window

Figure: The Histogram Window

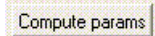


The **Histogram Window** shows a bar graph representing the mesh quality. The X axis represents element quality (usually normalized to between 0 and 1) and the Y axis represents the number of elements. Other functions which utilize this space will become pop-up menus if the quality or histogram is enabled.

The Data Entry Zone (DEZ)

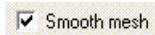
The DEZ provides access to the parameters associated with a particular operation. The controls utilized by the DEZ are described here:

Button



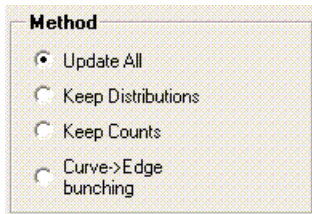
A button is used to perform a function indicated by the button label.

Check Box



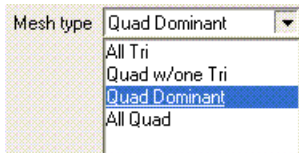
A check box is used to enable/disable an item or action indicated by the check box label.

Radio Buttons



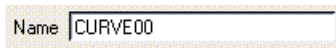
Radio buttons are a set of check boxes with the condition that only one can be enabled at a time. When you click the left mouse button on a radio button, it will be enabled, while all others will be disabled.

Drop-Down List



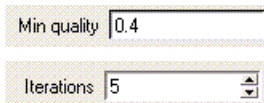
A drop-down list is a hidden single-selection list that shows only the current selection. Click the arrow button to display the list.

Text Entry



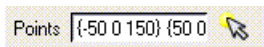
Text entries allow you to enter text associated with the label for the field.

Number Entry



Number entries allow you to enter numerical values for the parameter indicated by the label for the field. Some number entry fields may have arrow buttons which allow you to increase or decrease the value in entry field.

Selections



Selection fields indicate the entities selected for a particular operation. Click the button adjacent to the selection field to invoke the selection mode. The selection toolbar associated with the operation will appear. After confirming the selections, the selected items will be listed in the selection field.

Selection Toolbar

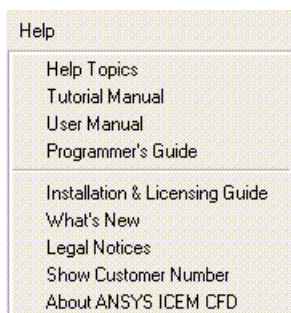


The selection toolbars contain some tools common to all select operations and some toggles for filtering entities for selection. Some controls are linked to the hotkeys available in the select mode.

Using the Help System

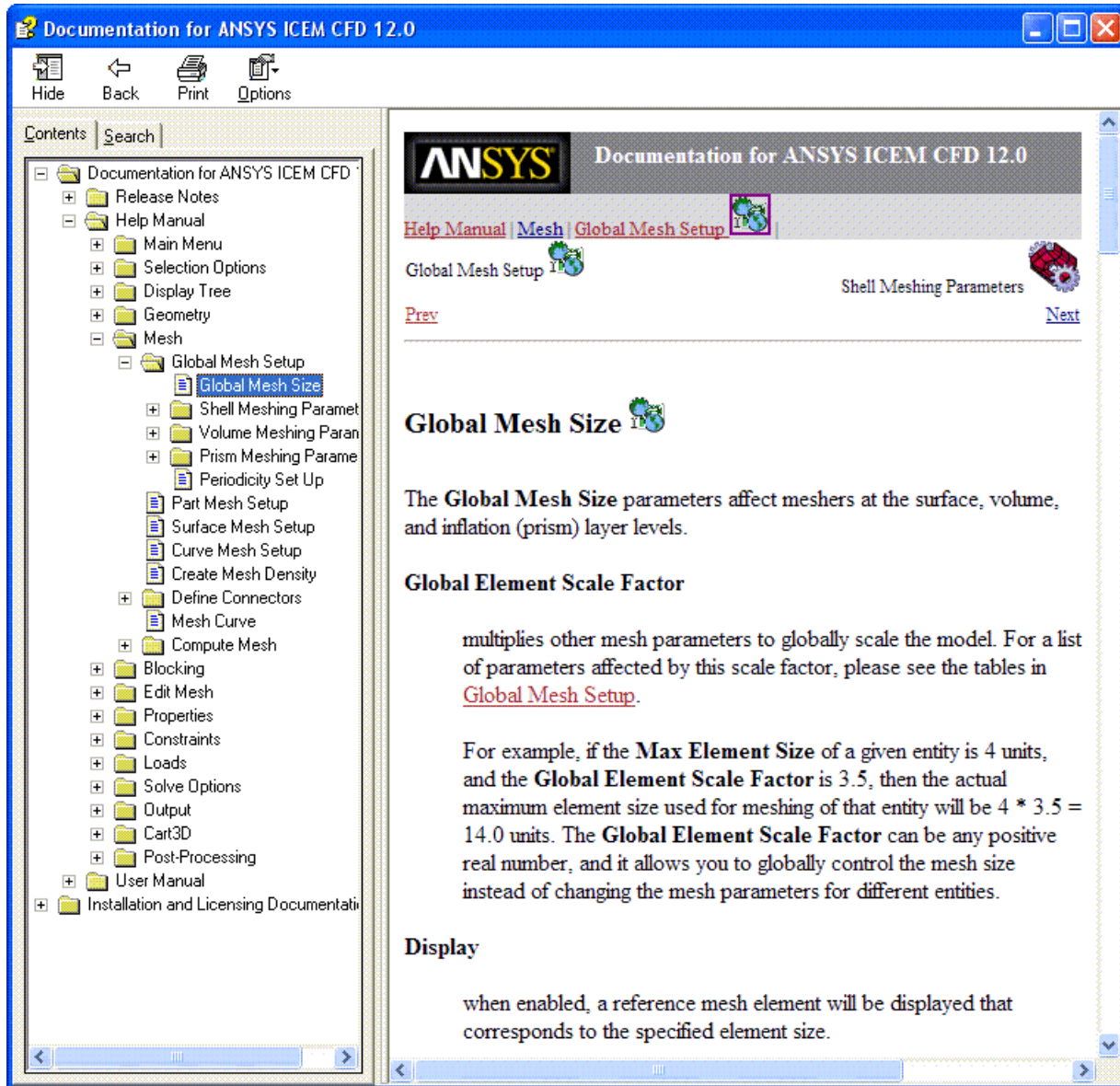
The product online help system provides easy access to the program documentation. The entire User's Manual, Help Manual and other documentation modules are available through the graphical user interface. Click **Help** on the main menu and select the appropriate option from the menu.

Figure: Help Menu Options



Online Help Interface

Figure: The Online Help Interface



- The help system is organized into different documentation modules which are further organized in sections, which are listed on the **Contents** tab. Click the document icon or topic title next to each section to display its content in the right windowpane.
- The **Search** tab allows you to view topics that contain certain words or phrases you specify. When you execute a search, all topics containing the search text display. To go to that topic, double-click the topic. To find out where you are in the help system, click the **Contents** tab. The highlighted entry in the table of contents indicates where the topic is.

The **Search** tab in the Windows Help includes several capabilities to assist you in narrowing down information returned in your searches. Some of these capabilities are:

- Using quotes to search for literal phrases.
- Using Boolean operators (AND, OR, NOT, NEAR) to precisely define search expressions.

- Using wildcard characters (*, ?) to search for expressions with identical characters.
- Using parentheses to nest search expressions.

The **Search** tab in the Help also includes check boxes located at the bottom of the panel that allow you to search previous results, match similar words, or search titles only.

CAD Repair

Before generating the mesh, you should confirm that the geometry is free of any flaws that would inhibit optimal mesh creation. If you wish to save the changes in the native CAD files, the following checks should be performed in a direct CAD interface:

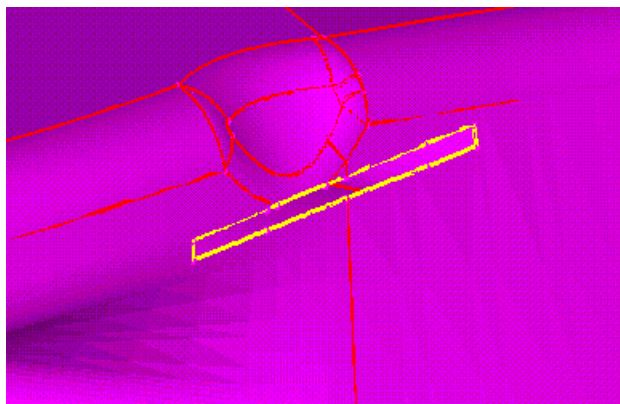
- To create a mesh, the **Tetra** mesher requires that the model contains a closed volume. If there are any holes (gaps or missing surfaces) in the geometry that are larger than the local tetras, the **Tetra** mesher will be unable to find a closed volume. Thus, if you notice any holes in the model prior to mesh generation, you should fix the surface data to eliminate these holes.
- The **Build Topology** operation will find holes and gaps in the geometry. It should display yellow curves where there are large (in relation to a user-specified tolerance) gaps or missing surfaces.
- During the Tetra process any leakage path (indicating a hole or gap in the model) will be indicated. The problem can either be corrected on a mesh level, or the geometry in that vicinity can be repaired and the meshing process repeated. For further information on the process of interactively closing holes, see the section Tetra > Tetra Generation Steps > Useful Region of Mesh.

For more information on CAD Repair topics, please go to [this page](#).

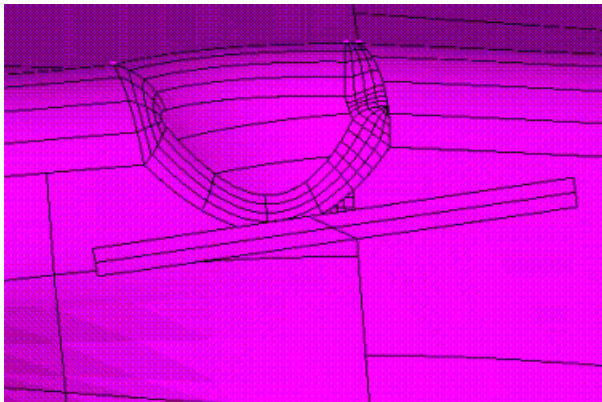
Close Holes

You can use the **Close Holes** option if the hole is bounded by more than one surface. For example, in [Figure : Hole Bounded by Multiple Surfaces \(p. 15\)](#), the yellow curves represent the boundary of the hole. It is clear that this hole is bounded by more than one surface.

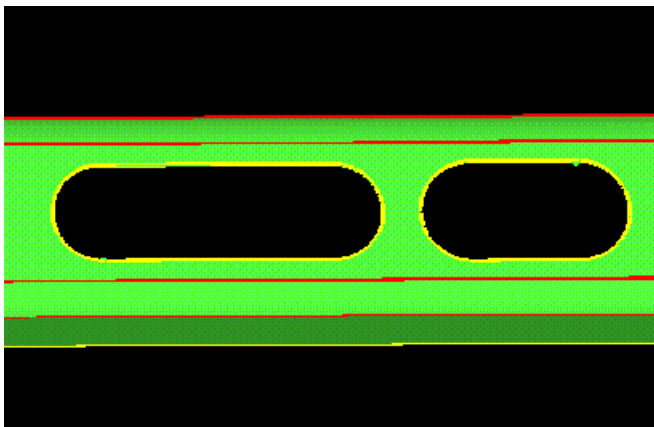
Figure: Hole Bounded by Multiple Surfaces



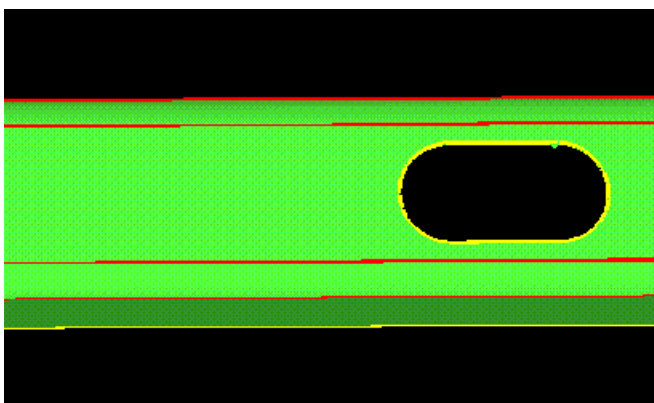
[Figure : Closed Hole \(p. 16\)](#) shows the geometry after the **Close Holes** operation is completed. A new surface is created to close the hole.

Figure: Closed Hole**Remove Holes**

You can use the **Remove Holes** option if the hole lies entirely within a single surface, such as a trimmed surface. For example, in [Figure : Hole Within a Single Surface \(p. 16\)](#), the two yellow curve loops represent the boundaries of the holes, which lie entirely in one surface.

Figure: Hole Within a Single Surface

[Figure : After Remove Holes \(p. 16\)](#) shows the geometry after the **Remove Holes** operation is completed for one of the holes. The existing surface is modified by removing the trim definition.

Figure: After Remove Holes

Fill, Trim and Blend in Stitch/Match Edges

Consider the case of a geometry with a gap shown in *Figure : Geometry With a Gap* (p. 17).

Figure: Geometry With a Gap

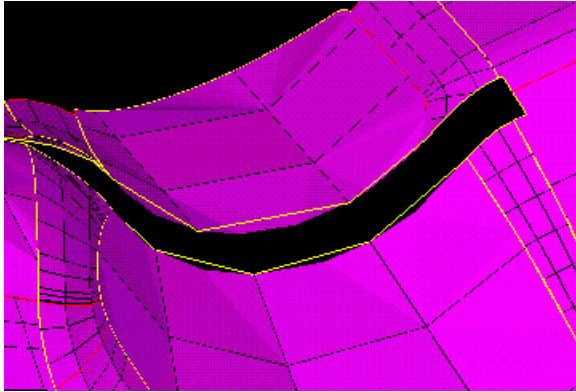


Figure : Using the Fill Option (p. 17) shows the use of the Fill option.

Figure: Using the Fill Option

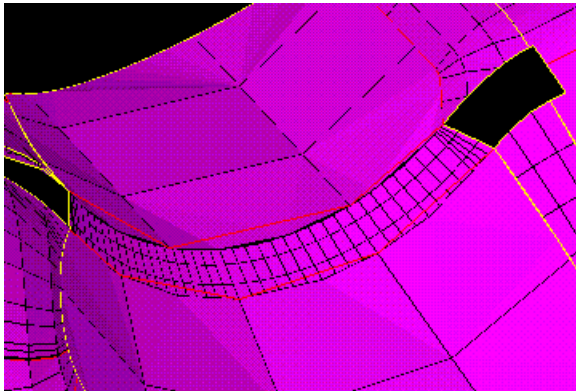


Figure : Using the Trim Option (p. 17) shows the use of the Trim option.

Figure: Using the Trim Option

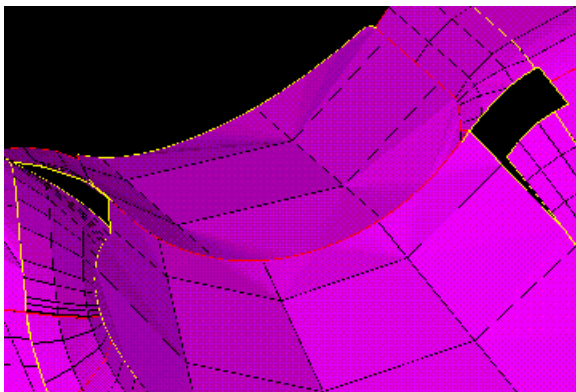
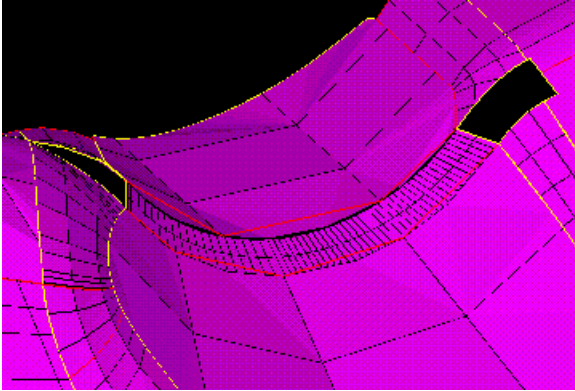


Figure : Using the Blend Option (p. 18) shows the use of the Blend option.

Figure: Using the Blend Option



Match in Stitch/Match Edges

The **Match** option is generally used in those cases where curves lie very close to each other, specifically when the two ends meet together (see [Figure : Geometry With Mismatched Edges](#) (p. 18) and [Figure : Geometry After Using the Match Edges Option](#) (p. 19)). You should have the two sets of curves within some tolerance for this option to work.

Figure: Geometry With Mismatched Edges

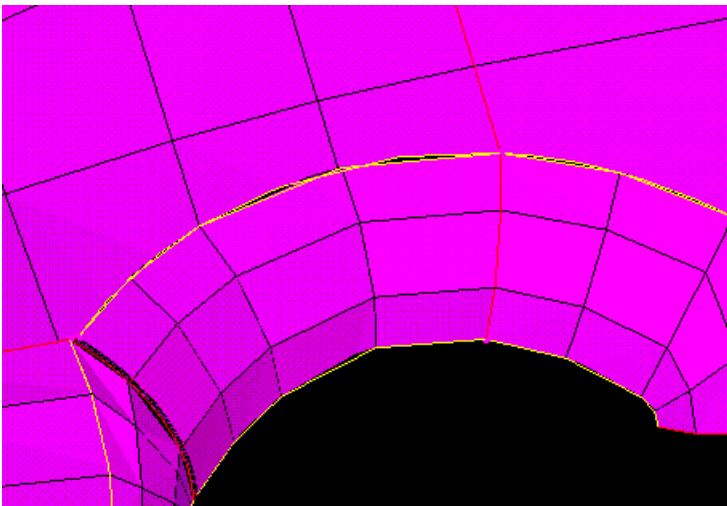
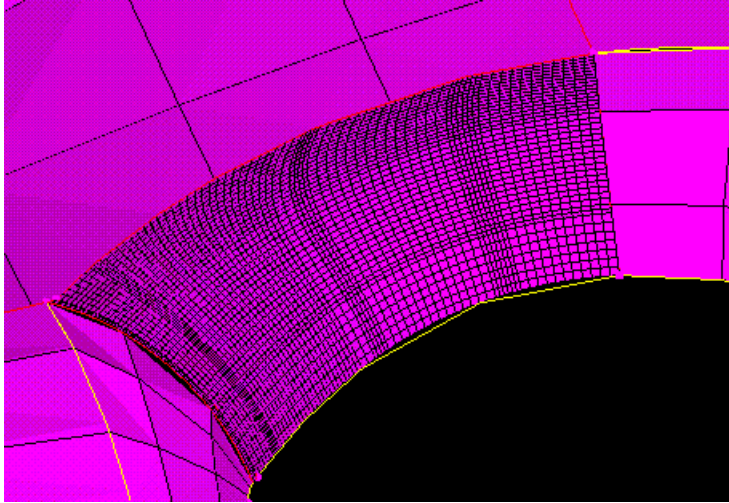


Figure: Geometry After Using the Match Edges Option



Tetra Meshing

Automated to the point that you have only to select the geometry to be meshed, the **Tetra** mesher generates tetrahedral meshes directly from the CAD geometry or STL data, without requiring an initial triangular surface mesh.

[Introduction](#)

[Tetra Generation Steps](#)

[Important Features in Tetra](#)

Introduction

The **Tetra** mesher can use different meshing algorithms to fill the volume with tetrahedral elements and to generate a surface mesh on the object surfaces. You can define prescribed curves and points to determine the positions of edges and vertices in the mesh. For improved element quality, the **Tetra** mesher incorporates a powerful smoothing algorithm, as well as tools for local adaptive mesh refinement and coarsening.

Tetra Mesh Generation

The **Tetra** mesher is suitable for complex geometries, and offers several advantages, including:

- Rapid model setup
- Mesh independent of underlying surface topology
- No surface mesh necessary
- Generation of mesh directly from CAD or STL surfaces
- Definition of element size on CAD or STL surfaces
- Control over element size inside a volume
- Nodes and edges of tetrahedra are matched to prescribed points and curves
- **Curvature/Proximity Based Refinement** automatically determines tetrahedra size for individual geometry features
- Volume and surface mesh smoothing, merging nodes and swapping edges
- Tetrahedral mesh can be merged into another tetra, hexa or hybrid mesh and then can be smoothed
- Coarsening of individual material domains
- Enforcement of mesh periodicity, both rotational and translational
- Surface mesh editing and diagnostic tools
- Local adaptive mesh refinement and coarsening
- One consistent mesh for multiple materials
- Fast algorithm: *1500 elements/second*
- Automatic detection of holes and easy way to repair the mesh
- For more details, go to [Run Tetra - The Octree Approach](#)

Input to Tetra

The following are possible inputs to the **Tetra** mesher:

- Sets of B-Spline curves and trimmed B-Spline surfaces with prescribed points
- Triangular surface meshes as geometry definition
- Full/partial surface meshes

B-Spline Curves and Surfaces

When the input is a set of B-Spline curves and surfaces with prescribed points, the mesher approximates the surface and curves with triangles and edges respectively; and then projects the vertices onto the prescribed points.

The B-Spline curves allow the **Tetra** mesher to follow discontinuities in surfaces. If no curves are specified at a surface boundary, the **Tetra** mesher will mesh triangles freely over the surface edge. Similarly, prescribed points allow the mesher to recognize sharp corners in the geometry. ANSYS ICEM CFD provides tools (**Build Topology**) to extract points and curves to define sharp features in the surface model.

Triangular Surface Meshes as Geometry Definition

Prescribed curves and points can also be extracted from triangulated surface geometry. This could be stereolithography (STL) data or a surface mesh converted to faceted geometry. Though the nodes of the **Tetra**-generated mesh will not exactly match the nodes of the given triangulated geometry, they will follow the overall shape. A geometry for meshing can contain both faceted and B-Spline geometry.

Full/Partial Surface Mesh

Existing surface mesh for all or part of the geometry can be specified as input to the **Tetra** mesher. The final mesh will then be consistent with and connected to the existing mesh nodes.

Tetra Generation Steps

The steps involved in generating a Tetra mesh are:

- Repairing/cleaning up the geometry
- Specifying geometry details
- Specifying sizes on surfaces/curves
- Meshing inside small angles or in small gaps between objects
- Desired mesh region
- Computing the mesh
- Checking the mesh for errors
- Editing the mesh to correct any errors
- Smoothing the mesh to improve quality

The mesh is then ready to apply loads, boundary conditions, etc., and for writing to the desired solver.

Repairing the Geometry

Refer to the [CAD Repair](#) section.

Geometry Details Required

In addition to a closed set of surfaces, the **Tetra** mesher requires curves and points where hard features (hard angles, corners) are to be captured in the mesh. *Figure : Curves and Points Representing Sharp Edges and Corners* (p. 23) shows a set of curves and points representing hard features of the geometry.

Figure: Curves and Points Representing Sharp Edges and Corners

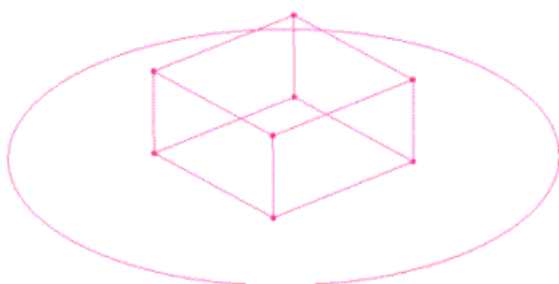


Figure : Mesh with Curves and Points (p. 23) shows the resultant surface mesh if the curves and points are preserved in the geometry. Mesh nodes are forced to lie along the curves and points to capture the hard features of the geometry.

Figure: Mesh with Curves and Points

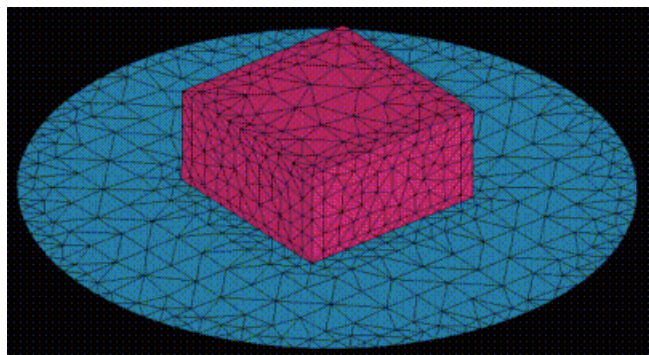
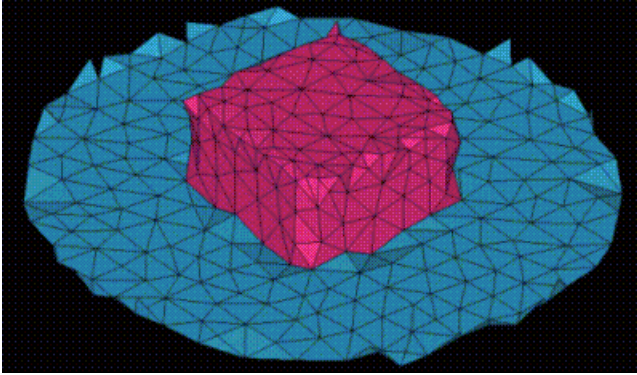


Figure : Mesh Without Curves and Points (p. 24) shows the resultant surface mesh if the curves and points are deleted from the geometry. The hard features of the geometry are not preserved, but rather are neglected or chamfered. The boundary mesh nodes lie on the surfaces, but they will only lie on the edges of the surfaces if curves and points are present. Removal of curves and points can be used as a geometry defeaturing tool.

Figure: Mesh Without Curves and Points

Sizes on Surfaces and Curves

To produce the optimal mesh, it is essential that all surfaces and curves have the proper tetra sizes assigned to them. For a visual representation of the mesh size, select **Geometry > Surfaces > Tetra Sizes** from the **Display Tree**. The same can be done with **Curves**. Tetra icons will appear, representing the element size of the mesh to be created on these entities. Using the mouse, you may rotate the model and visually confirm that the tetra sizes are appropriate. If a curve or surface does not have an icon plotted on it, the icon may be too large or too small to see. In this case, modify the mesh parameters so that the icons are visible in a normal display.

To modify the mesh size for all entities, adjust the **Scale Factor**, which is found under **Mesh > Global Mesh Setup**. Note that if the **Scale Factor** is assigned a value of 0, the **Tetra** mesher will not run.

Meshing Inside Small Angles or in Small Gaps Between Objects

Examine the regions between two surfaces or curves that are very close together or that meet at a small angle. (This would also apply if the region outside the geometry has small angles.) If the local tetra sizes are not small enough so that at least 1 or 2 elements would fit through the thickness, you should define thin cuts. This option is in the **Mesh > Global Mesh Setup > Volume Meshing Parameters** section. To define a thin cut, the two surfaces have to be in different Parts. If the surfaces meet, the curve at the intersection of the surfaces will need to be in a different part.

If the tetra sizes are larger or approximately the same size as the gap between the surfaces or curves, the surface mesh could have a tendency to jump the gap, thus creating non-manifold vertices. These non-manifold vertices would be created during the meshing process. The **Tetra** mesher automatically attempts to close all holes in a model. Since the gap may be confused as a hole, you should either define a thin cut, in order to establish that the gap is not a hole; or make the mesh size small enough so that it won't close the gap when the meshing is performed. A hole is usually considered a space that is greater than 2 or 3 elements in thickness.

Desired Mesh Region

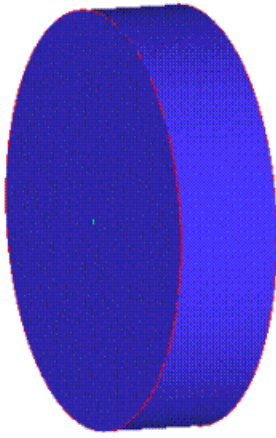
During the process of finding the bounding surfaces to close the volume mesh, the mesher will determine if there are holes in the model. If holes are found, the Message window will display a message like "Material point ORFN can reach material point LIVE." You will be prompted with a dialog box saying, "Your geometry has a hole, do you want to repair it?" A jagged line will display the leakage path from the ORFN part to the LIVE part. The elements surrounding the hole will also be displayed. To repair the hole, select the single edges bounding it - and the mesher will loft a surface mesh to close the hole. Further holes would be flagged

and repaired in the same manner. If there are many problem areas, it may be better to repair the geometry or adjust the meshing parameters.

The Octree Mesh Method

The Octree mesh method is based on the following spatial subdivision algorithm: This algorithm ensures refinement of the mesh where necessary, but maintains larger elements where possible, allowing for faster computation. Once the "root" tetrahedron, which encloses the entire geometry, has been initialized, **Tetra** subdivides the root tetrahedron until all element size requirements are met.

Figure: Geometry Input to Tetra



At this point, the Tetra mesher balances the mesh so that elements sharing an edge or face do not differ in size by more than a factor of 2.

Figure: Full Tetra Enclosing the Geometry

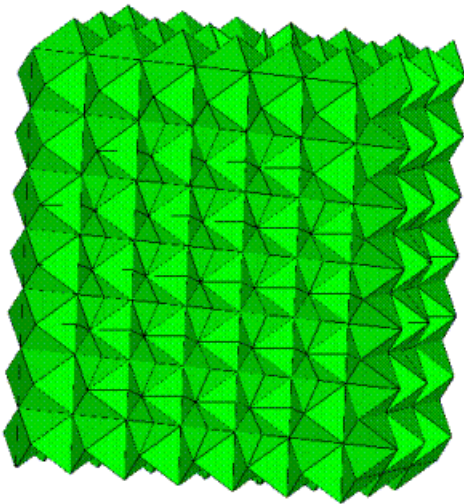
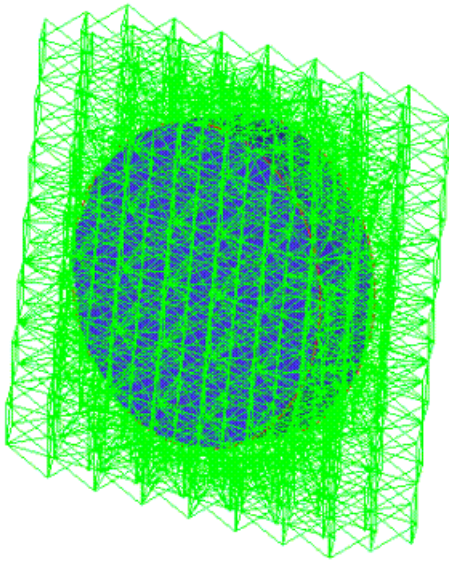
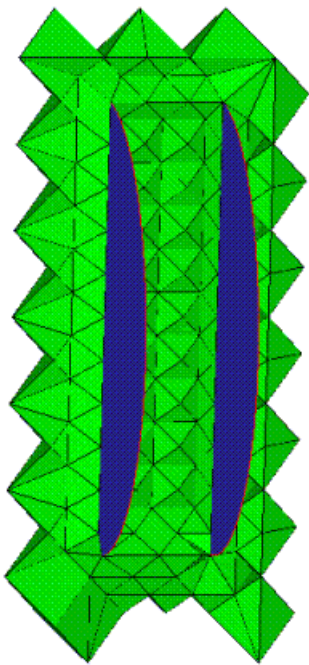


Figure: Full Tetra Enclosing the Geometry in Wire Frame Mode**Figure: Cross-Section of the Tetra**

After this is done, Tetra makes the mesh conformal - that is, it guarantees that each pair of adjacent elements will share an entire face. The mesh does not yet match the given geometry, so the mesher next rounds the nodes of the mesh to the prescribed points, prescribed curves or model surfaces. Tetra then "cuts away" all of the mesh, which cannot be reached by a user-defined material point without intersection of a surface.

Figure: Mesh after it captures surfaces and separation of useful volume

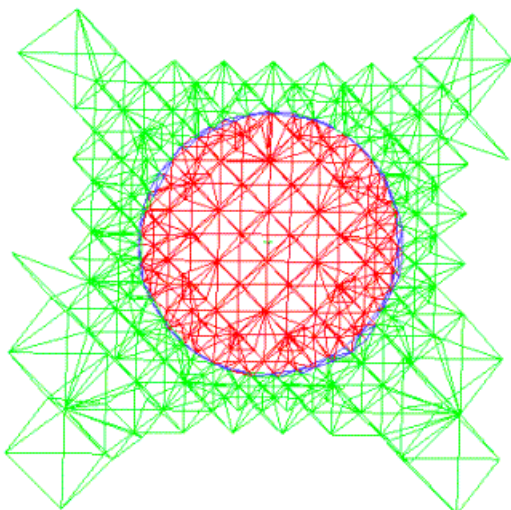
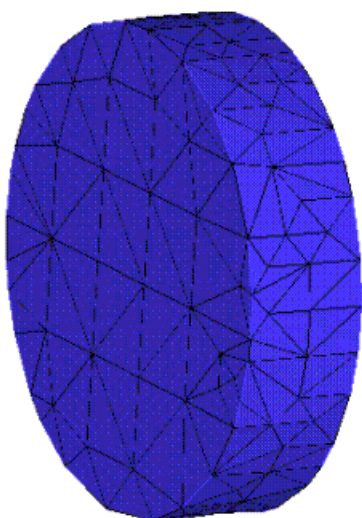
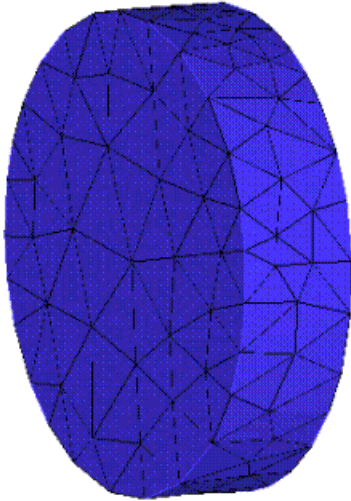


Figure: Final Mesh before smoothing



Finally, the mesh is smoothed by moving nodes, merging nodes, swapping edges and in some cases, deleting bad elements.

Figure: Final Mesh after smoothing

Important Features in Tetra

The following sections describe important features of tetra meshing.

Curvature/Proximity Based Refinement

If the maximum tetrahedral size defined on a surface is larger than needed to resolve the feature, you can employ **Curvature/Proximity Based Refinement** to automatically subdivide the mesh to capture the feature. The value specified is proportional to the global scale factor, and is the smallest size to be achieved through automatic element subdivision. Even with large sizes specified on the surfaces, the features can be captured automatically.

The **Curvature/Proximity Based Refinement** value is the minimum element size to be achieved via automatic subdivision. If the maximum size on a geometry entity is smaller than the **Curvature/Proximity Based Refinement** value, the Tetra mesher will still subdivide to meet that requested size. The effect is a geometry-based adaptation of the mesh.

Tetrahedral Mesh Smoother

In smoothing the mesh, the tetrahedral smoother calculates individual element quality based on the selection from the list of available criteria.

The smoother modifies the elements with quality below the specified **Up to quality** value. Nodes can be moved and/or merged, edges are swapped, and in some cases elements are deleted. This operation is then repeated on the improved grid, up to the specified number of iterations. You can choose to smooth some element types while freezing others.

Tetrahedral Mesh Coarsener

During the coarsening process you can exclude surface or material domains. If the **Maintain surface sizes** option is enabled during coarsening, the resulting mesh satisfies the specified mesh size criteria on the geometric entities.

Triangular Surface Mesh Smoother

The triangular surface mesh inherent in the **Tetra** mesh generation process can also be used independently of the volume mesh. The triangular smoother marks all elements that are initially below the quality criterion and then runs the specified number of smoothing steps on the elements. Nodes are moved on the actual CAD surfaces to improve the quality of the elements.

Triangular Surface Mesh Coarsener

In the interest of minimizing grid points, the coarsener reduces the number of triangles in a mesh by merging triangles. This operation is based on the maximum deviation of the resultant triangle center from the surface, the aspect ratio of the merged triangle and the maximum size of the merged triangle.

Triangular Surface Editing Tools

There are tools available under the **Edit Mesh** menu for interactive mesh editing, where nodes can be moved on the underlying CAD surfaces, merged or even deleted. Individual triangles of the mesh can be subdivided or tagged with different names. You can perform quality checks, as well as local smoothing.

Diagnostic tools for surface meshes allow you to fill holes easily in the surface mesh. Also there are tools for the detection of overlapping triangles and non-manifold vertices, as well as detection of single/multiple edge and duplicate elements.

Check Mesh

Check the validity of the mesh using **Edit Mesh > Check Mesh**.

You can opt to use the **Create subsets** option for each of the problems so that they can be fixed later or can opt to use the **Check/fix each** option to check and fix each one of them. Using subset manipulation and mesh editing techniques, you can diagnose the problem and resolve it through merging nodes, splitting edges, swapping edges, delete/create elements, etc.

For ease of use when working with subsets, it is usually helpful to add elements to the subset in order to see what is happening around the problem elements. To do this, right-click on the Subset name in the Display tree and then add layers of elements to the subset. It is also useful to display the element nodes and/or display the elements slightly smaller than actual size. Both of these options can be accessed by right-clicking on Mesh in the Display Tree.

Keep in mind that after mesh editing, the diagnostics should be re-checked to verify that no mistakes were made.

There are several checks for **Errors** as well as **Possible problems**. The descriptions of each of these checks can be found in the **Edit Mesh > Check Mesh** section of the Help Manual.

Smooth Mesh Globally

After eliminating errors/possible problems from a tetra mesh, you need to smooth the grid using **Edit Mesh > Smooth Mesh Globally** to improve the quality.

Smoothing iterations

This value is the number of times the smoothing process will be performed. Models with a more complicated geometry will require a greater number of iterations to obtain the desired quality, which is specified for **Up to quality**.

Up to quality

The **Min** value represents the worst quality, while the **Max** value represents the highest quality elements. Usually, the **Min** value is set to 0.0 and the **Max** value is set to 1.0. The **Up to quality** value gives the smoother a quality to aim for. Ideally, after smoothing, the quality of the elements should be higher than or equal to this value. If this does not happen, you should employ other methods of improving the quality, such as merging nodes and splitting edges. For most models, the elements should all have ratios of greater than 0.3, while a ratio of 0.15 for complicated models is usually sufficient.

Freeze

If the **Freeze** option is selected for an element type, the nodes of this element type will be fixed during the smoothing operation. As a result, this element type will not be displayed in the histogram.

Float

If the **Float** option is selected, the nodes of the specified element type will be capable of moving freely, allowing nodes that are common with another type of element to be smoothed. The quality of elements set to float is not tracked during the smoothing process and so the quality is not displayed in the histogram.

The tetrahedral quality will be displayed within the **Quality Histogram**, where 0 represents the worst aspect ratio and 1 represents the best aspect ratio. You may modify the display of the histogram by adjusting the values of **Min**, **Max**, **Height**, and **Bars**. These options as well as the following options can be accessed by right-clicking on the histogram.

- The **Replot** option allows you to change the following parameters in the window that will appear. Clicking **Accept** will replot the histogram to the newly set values.

Min X Value

This minimum value, which is located on the left-most side of the histogram's x-axis, represents the worst quality elements.

Max X Value

This maximum value, which is located on the right-most side of the histogram's x-axis, represents the highest quality that elements can achieve.

Max Y height

You can adjust the number of elements that will be represented on the histogram's y-axis. Usually a value of 20 is sufficient. If there are too many elements displayed, it is difficult to discern the effects of smoothing.

Num bars

This represents the number of subdivisions within the range between the **Min** and the **Max** values. The default **Bars** have widths of 0.05. Increasing the number of displayed bars, however, will decrease this width.

- The **Reset** option will return all of the values back to the original parameters.
- **Show**: Click the left mouse button on any of the bars in the histogram and the color will change from green to pink. Enabling **Show** will display the elements that fall within the selected range on the model in the main viewing window.
- **Solid**: Enabling this option will display the elements as solid, rather than as the default grid representation. You need to enable **Show**, as well, to activate this option.
- **Subset**: If you highlight bars from the histogram and enable **Show**, the elements displayed in white color will be placed into a **Subset**. The visibility of this subset is controlled by **Subset** in the **Display Tree**. The **Add select** option allows you to add elements to an already established subset.

Quality Metrics

This option allows you to modify the histogram display.

The histogram displays the overall quality of the mesh. The x-axis measures the quality, with 0 representing poor quality and 1 representing high quality. The y-axis measures the number of elements that belong within each quality sub-range.

For descriptions of all the quality metrics, refer to the [Edit Mesh > Display Mesh Quality](#) section in the Help Manual.

Advanced Options for Smoothing Mesh

Prism Warpage Ratio

Prisms are smoothed based on a balance between prism warpage and prism aspect ratio. Values from 0.01 to 0.50 favor improving the prism aspect ratio, while those from 0.50 to 0.99 favor improving prism warpage. A value of 0.5 favors neither. The farther the value is from 0.5, the greater the effect.

Stay on geometry

The default is, when a grid is smoothed, the nodes are restricted to the geometry -- surface, curves and points -- and can only be moved along the geometrical entities to obtain a better mesh.

Violate Geometry

Enabling this option allows the smoothing operation to yield a higher quality mesh by violating the constraints of the geometry. The nodes can be moved off of the geometry to obtain better mesh quality, as long as the movement remains within the absolute distance specified.

Relative Tolerance

This option works in the similar fashion as **Violate Geometry** except that the distance is relative here.

Allow refinement

If the quality of the mesh cannot be improved through normal algebraic smoothing, the **Allow refinement** option will allow the smoother to automatically subdivide elements to obtain further improvement. After smoothing with **Allow refinement** enabled, it may be necessary to smooth further with the option disabled. The goal is to reduce the number of elements that are attached to one vertex by refinement in problem regions.

Laplace smoothing

This option will solve the Laplace equation, which will generally yield a more uniformly spaced mesh.

Note

This can sometimes lead to a lower determinant quality of the prisms. Also, this option works only for the triangular surface mesh.

Allow node merging

This option will collapse and remove the worst tetra and prism elements when smoothing in order to obtain a higher quality mesh. This is enabled by default, and is often very useful in improving the grid quality.

Not just worst 1%

This option will smooth all of the geometry's elements to the assigned quality (specified under **Up to quality**) not just focus on the worst 1% of the mesh. Typically, when a mesh is smoothed, the smoother concentrates on improving the worst regions; this option will allow the smoother to continue smoothing beyond the worst regions until the desired quality is obtained.

Surface fitting

This option will smooth mesh, keeping the nodes and the new mesh restricted along the surface of the geometry. Only **Hexa** models will utilize this option.

Ignore PrePoints

This option will allow the smoother to attempt to improve the mesh quality without being bound by the initial points of the geometry. This option is similar to the **Violate geometry** option, but works only for points located on the geometry. This option is available only when there are hexahedral elements in the model. Usually, the best way to improve the quality of grids that cannot be smoothed above a certain level is to concentrate on the surface mesh near the bad elements and edit this surface mesh to improve the quality.

Prism Mesh

Tetra meshing is not efficient for capturing shear or boundary layer physics. Prism mesh efficiently captures these effects near the surface while maintaining the ease and automation of Tetra mesh. Prism has always been necessary for CFD customers, but now that the option is more widely available, many other branches of CAE have started using prisms to better resolve the physics perpendicular to the surfaces of their models. With ANSYS ICEM CFD, Prism and Tetra generation is automatic and intelligent.

The spacing of the prism layers to capture the Y^+ for Navier-Stokes mesh is the primary concern. The rate of volume change between cells is also important. Calculations are done between nodes or elements, and Prism mesh gives you more elements perpendicular to the surface. This efficiently allows for better resolution (more calculations per unit distance) of the solution normal to the surface, without increasing the number of elements along the surface. This gives you a quicker and more accurate solution than a very fine tetra mesh.

The height and direction of the prism layer extrusion are calculated on an element by element basis and may vary due to global or local controls, or for improved quality. You may want to set the initial height, number of layers and growth ratio, and then they limit this with the Prism height limit factor. Or you may prefer to just set the number of layers and growth ratio. This allows Prism to adjust the initial height and locally optimize the volume transition between the prisms and tetras. Users concerned about Y^+ can then adjust the first cell height using **Edit Mesh > Split Mesh > Split Prisms**.

Prism parameters are set globally, but can then be adjusted on a part by part or entity by entity basis. Entity settings override global settings and between entities, the smaller size overrides the larger. For instance, 3 layers could be set for a growth rate of 1.2 globally, but a certain part could be set for 5 layers. On a specific entity within that part or another part, you could set a specific parameter. If you set prism parameters, such as height, on a curve entities, it will interpolate that parameter across the surface between the curves.

You may notice that you can also select volume parts for prism. If no volume parts are selected, it will assume that you want to grow prisms into all volumes bordering the prism surfaces. If you select specific volume parts, then prism will only be grown into those volumes.

After each layer is extruded, smoothing is done according to the global settings. The layers are grown one at a time. This continues until all the requested layers are grown. You can add prisms to existing layers or you can subdivide and redistribute layers at a later date. You can save time by growing only a few thicker layers and then subdividing them into many layers. The smoothing is the most time consuming part, so for simple configurations, it may be best to turn off all smoothing but grow all the layers one at a time. This allows you to take advantage of the variable height feature.

There are two ways to generate prism mesh:

Compute Mesh > Prism Mesh

This option is used to grow prism layers next to wall geometries. You can define the local initial height, growth ratio, and number of layers at Mesh > Global Mesh Setup > Global Prism Settings. This option can create prisms from existing volume or surface mesh.

Note

If the existing volume mesh is tet/hexa mesh, on the hexa side the prisms will be added within the first hexa layer.

Compute Mesh > Volume Mesh > Create Prism Layers

This option allows you to directly create tetra mesh with prisms next to wall geometries. You can choose whether to create prism mesh from the geometry and/or the surface mesh.

Note

If many prism layers are created, it is faster and can be more robust to create initial prism layers and then split them to create the total desired number of prism layers using Edit Mesh > Split Mesh > Split Prisms.

For more useful information on Prism Mesh, please go to [this page](#).

Prism Mesh Process

The prism mesh process generates prism elements near boundary surfaces from tetrahedral or tri surface mesh. This batch process creates prisms by extrusion of the surface mesh, and the resulting prisms are made conformal with any existing tetrahedral volume mesh. The prism mesh can be smoothed to yield the necessary quality.

Prism Mesh Preparation

When generating prism mesh, preparation is key. It is easier to edit a tetra mesh than a tetra prism mesh. Prism mesh can also be difficult to smooth, so it will save time to start with good quality tetra or tri surface mesh.

Start with the best possible initial hybrid mesh quality

Hybrid mesh is generally difficult to smooth.

Start with good Tetra or Tri surface mesh

- Choose prism options carefully
- Check aspect ratios / quality.
- Check and fix all diagnostics. Single/multiple edges, Non-manifold vertices, and Duplicate elements will crash the prism mesher.
- Visually scan the surface mesh. Look for any surface discrepancies or sharp tent-like structures in the mesh.
- Make sure part associations are correct.

Look for a few elements of one part scattered among another part. Extruding from a few isolated elements will likely crash the prism mesher. Modify part assignments of such elements.

- Use the Smoothing Options for Tetra and Tri surface mesh under Mesh > Prism before creating prism mesh.
- Laplace Triangle Quality type is typically best for eventual prism quality.

Smoothing Tetra/Prism Mesh

First smooth the tetra and tri elements (set PENTA_6 to **Freeze**). Once the tetra and tri elements are as smooth as possible, then smooth all elements at the same time (including PENTA_6). Decrease the **Up to quality** value, so that the prism elements are not distorted too much.

Hexa

Hexa is a 3-D object-based, semi-automatic, multi-block structured and unstructured, surface and volume mesher.

Introduction

Hexa represents a new approach to hexahedral mesh generation. The block topology model is generated directly on the underlying CAD geometry. Within an easy-to-use interface, those operations most often performed by experts are readily accessible through automated features.

There is access to two types of entities during the mesh generation process in **Hexa**: block topology and geometry. After interactively creating a 3-D block topology model equivalent to the geometry, the block topology may be further refined through the splitting of edges, faces and blocks. In addition, there are tools for moving the block vertices -- individually or in groups -- onto associated curves or CAD surfaces. You may also associate specific block edges with important CAD curves to capture important geometric features in the mesh.

Moreover, for models where you can take advantage of symmetry conditions, topology transformations such as translate, rotate, mirror and scaling are available. The simplified block topology concept allows rapid generation and manipulation of the block structure and, ultimately, rapid generation of the hexahedral meshes.

Hexa provides a projection-based mesh generation environment where, by default, all block faces between different materials are projected to the closest CAD surfaces. Block faces within the same material may also be associated to specific CAD surfaces to allow for definition of internal walls. In general, there is no need to perform any individual face associations to underlying CAD geometry, which further reduces the difficulty of mesh generation.

For more useful information on Hexa, please go to [this page](#).

Features of Hexa

Some of the more advanced features of **Hexa** include:

O-grids: For very complex geometry, **Hexa** automatically generates body-fitted internal and external O-grids to parametrically fit the block topology to the geometry to ensure good quality meshes.

Edge-Meshing Parameters: Hexa's edge-meshing parameters offer unlimited flexibility in applying user specified bunching requirements.

Time Saving Methods: Hexa provides time saving surface smoothing and volume relaxation algorithms on the generated mesh.

Mesh Quality Checking: With a set of tools for mesh quality checking, elements with undesirable skewness or angles may be displayed to highlight the block topology region where the individual blocks need to be adjusted.

Mesh Refinement/Coarsening: Refinement or coarsening of the mesh may be specified for any block region to allow a finer or coarser mesh definition in areas of high or low gradients, respectively.

Replay Option: Replay file functionality enables parametric block topology generation linked to parametric changes in geometry.

Symmetry: As necessary in analyzing rotating machinery applications, for example, **Hexa** allows you to take advantage of symmetry in meshing a section of the rotating machinery thereby minimizing the model size.

Link Shape: This allows you to link the edge shape to existing deforming edge. This gives better control over the grid specifically in the case of parametric studies.

Adjustability: Options to generate 3-D surface meshes from the 3-D volume mesh and 2-D to 3-D block topology transformation.

Mesh Generation with Hexa

To generate a mesh within **Hexa** you need to:

- Import a geometry file using any of the direct, indirect or faceted data interfaces.
- Interactively define the block model through split, merge, O- grid definition, edge/face modifications and vertex movements.
- Check the block quality to ensure that the block model meets specified quality thresholds.
- Assign edge meshing parameters such as maximum element size, initial element height at the boundaries and expansion ratios.
- Generate the mesh with or without projection parameters specified. Check **Mesh quality** to ensure that specified mesh quality criteria are met.
- Write **Output** files to the desired solvers.

If necessary, you may always return to previous steps to manipulate the blocking if the mesh quality does not meet the specified threshold or if the mesh does not capture certain geometry features. The blocking may be saved at any time, thus allowing you to return to previous block topologies.

Additionally, at any point in this process, you can generate the mesh with various projection schemes such as full face projection, edge projection, point projection or no projection at all.

Note

In the case of no projection, the mesh will be generated on the faces of the block model and may be used to quickly determine if the current blocking strategy is adequate or if it must be modified.

The Hexa Database

The Hexa database contains both geometry and block topology data, each containing several sub-entities.

The Geometric Data Entities:

- **Points:** x, y, z point definition
- **Curves:** trimmed or untrimmed NURBS curves

- **Surfaces:** NURBS surfaces, trimmed NURBS surfaces

The Block Topologic Data Entities:

- **Vertices:** corner points of blocks, of which there are at least eight, that define a block
- **Edges:** a face has four edges and a block twelve
- **Faces:** six faces make up a block
- **Blocks:** volume made up of vertices, edges and faces

Intelligent Geometry in Hexa

Using **ANSYS ICEM CFD's** Direct CAD Interfaces, which maintain the parametric description of the geometry throughout the CAD model and the grid generation process, hexahedral grids can be easily remeshed on the modified geometry.

The geometry is selected in the CAD system and tagged with information (made intelligent) for grid generation such as boundary conditions and grid sizes and this intelligent geometry information is saved with the master geometry.

In **Hexa**, by updating all entities with the update projection function, blocking vertices projected to prescribed points in the geometry are automatically adapted to the parametric change and one can recalculate the mesh immediately. Additionally, with the use of its **Replay** functionality, **Hexa** provides complete access to previous operations.

Unstructured and Multi-block Structured Meshes

The mesh output of **Hexa** can be either unstructured or multi-block structured and need not be determined until after you have finished the whole meshing process when the output option is selected.

Unstructured Mesh Output

The unstructured mesh output option will produce a single mesh output file where all common nodes on the block interfaces are merged, independent of the number of blocks in the model.

Multi-Block Structured Mesh Output

Used for solvers that accept multi-block structured meshes, the multi-block structured mesh output option will produce a mesh output file for every block in the topology model.

For example, if the block model has 55 blocks, there will be 55 output files created in the output directory.

Additionally, without merging any of the nodes at the block interfaces, the Output Block option allows you to minimize the number of output files generated with the multi-block structured approach.

Blocking Strategy

With **Hexa**, the basic steps necessary to generate a hexahedral model are the same, regardless of model complexity. The blocking topology, once initialized, can then be modified by splitting and merging the blocks, as well as through the use of an operation called **O-grid** (Refer to the next section). While these operations are performed directly on the blocks, the blocks may also go through indirect modification by altering the sub-entities of the blocks (i.e.: the vertices, edges, faces).

Upon initialization, **Hexa** creates one block that encompasses the entire geometry. The subsequent operations under the **Blocking** menu of developing the block model, referred to as "blocking the geometry," may be performed on a single block or across several blocks.

Note

Note : The topologic entities in Hexa are color-coded based on their properties.

Colors of Edges:

White Edges and Vertices

These edges are between two material volumes. The edge and the associated vertices will be projected to the closest CAD surface between these material volumes. The vertices of these edges can only move on the surfaces.

Blue Edges and Vertices

These edges are in the volume. The vertices of these edges, also blue, can be moved by selecting the edge just before it and can be dragged on that edge.

Green Edges and Vertices

These edges and the associated vertices are being projected to curves. The vertices can only be moved on the curves to which it is being projected.

Red Vertices






These vertices are projected to prescribed points.

Hexa Block Types

When blocking a model, it is important to note that the block type affects many operations within Hexa and the entire approach to mesh generation. For example, if you split a model with mapped blocks, the split will propagate through faces that have a mapped relationship on the opposite side. For free blocks, a split will terminate at the free face. Similarly, if you set edge parameters on a mapped face edge, opposite edges will have a similar number of nodes. If however, that edge is attached to a free face, the number of nodes on the opposite side will not be adjusted. Using this free/mapped relationship, you can shape the blocking and resulting mesh.

The ability to convert blocks from free to mapped or vice versa imposes constraints on the blocking and resulting mesh. By imposing more constraints, you can enforce a greater number of hexa elements, while reducing the constraints can sometimes improve mesh transitioning.

Figure: Hexa Block Types

Dimension	3D	2D
Mapped / Structured	 i, j, k are mapped	 i, j are mapped
Swept	 j is mapped	N/A
Free/Unstructured	 All edges are free from mapping	 All edges are free from mapping

Split

The **Split** function, which divides the selected block interactively, may be applied across the entire block or to an individual face or edge of a block by using the **Split face** or **Split edge** options, respectively. Blocks may be isolated using the **Index control**.

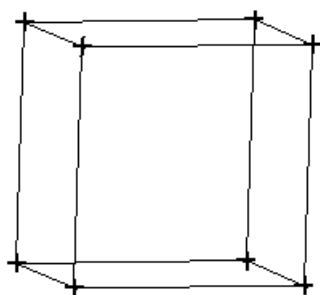
Merge

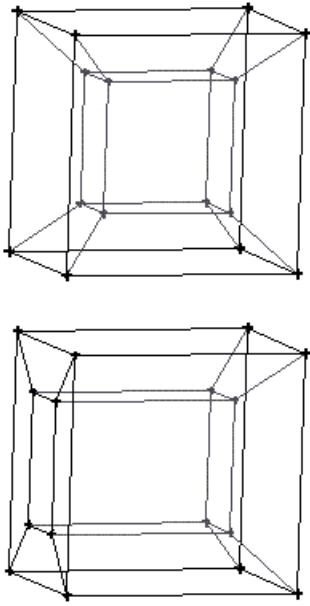
The **Merge** function works similar to split blocks; one can either merge the whole block or merge only a face or an edge of the block.

While some models require a high degree of blocking skill to generate the block topology, the block topology tools in **Hexa** allow you to quickly become proficient in generating a complex block model.

Using the Automatic O-grid

The **O-grid** creation capability is simply the modification of a single block or blocks to a 5 sub-block topology as shown below. There are several variations of the basic **O-grid** generation technique and the **O-grid** shown below is created entirely inside the selected block.

Figure: Initial block, block with O-Grid, O-Grid with include a face



Using the **Add face** option, an **O-grid** may also be created such that the **O-grid** passes through the selected block faces. In Figure , the **Add Face** option was used on the right most block to add the bottom face on the block prior to generating the **O-grid**.

Another important feature of the automatic **O-grid** is the ability to re-scale the **O-grid** after generation. When the **O-grid** is generated, the size of the **O-grid** is scaled based upon a factor in the **Blocking > O-grid parameter** window. The **Re scale O grid** option allows you to re-scale the previously generated **O-grid**.

The blocks may also be modified by moving the vertices of the blocks and by defining specific relationships between the faces, edges and vertices to the geometry.

Most Important Features of Hexa

Hexa has emerged as the quickest and most comprehensive software for generating large, highly accurate, 3D-geometry based hexahedral meshes. Now, in the latest version of Hexa, it is also possible to generate 3D surface meshes with the same speed and flexibility.

- CAD- and projection-based hexahedral mesh generation
- Easy manipulation of the 3D object-based topology model
- Modern GUI and software architecture with the latest hexahedral mesh technology
- Extensive solver interface library with over 100 different supported interfaces
- Automatic O-grid generation and O-grid re-scaling
- Geometry-based mesh size and boundary condition definition
- Mesh refinement to provide adequate mesh size in areas of high or low gradients
- Smoothing/relaxation algorithms to quickly yield quality meshes
- Generation of multi-block structured, unstructured, and super- domain meshes
- Ability to specify periodic definitions
- Extensive replay functionality with no user interaction for parametric studies

- Extensive selection of mesh bunching laws including the ability to graphically add/delete/modify control points defining the graph of the mesh bunching functions
- Link bunching relationships between block edges to automate bunching task
- Topology operations such as translate, rotate, mirror, and scaling to simplify generation of the topology model
- Automatic conversion of 3D volume block topology to 3D surface mesh topology
- Automatic conversion of 2D block topology to 3D block topology
- Block face extrusion to create extended 3D block topology
- Multiple projection options for initial or final mesh computation
- Quality checks for determinant, internal angle and volume of the meshes
- Domain renumbering of the block topology
- Output block definition to reduce the number of multi-block structured output mesh files
- Block orientation and origin modification options

Automatic O-grid generation

Generating **O-grids** is a very powerful and quick technique used to achieve a quality mesh. This process would not have been possible without the presence of O-grids. The O-grid technique is utilized to model geometry when you desire a circular or "O"-type mesh either around a localized geometric feature or globally around an object.

Important Features of an O-grid

Generation of Orthogonal Mesh Lines at an Object Boundary

The generation of the O-grid is fully automatic and you simply select the blocks needed for O-grid generation. The O-grid is then generated either inside or outside the selected blocks. The O-grid may be fully contained within its selected region, or it may pass through any of the selected block faces.

Rescaling an O-grid After Generation

When the O-grid is generated, the size of the O-grid is scaled based upon the **Factor** in the **Blocking > O-grid** parameter window. You may modify the length of the O-grid using the **Blocking > Re-scale O-grid** option. If a value that is less than 1 is assigned, the resulting O-grid will be smaller than the original. If, however, a value is larger than 1, the resulting O-grid will be larger.

Edge Meshing Parameters

The edge meshing parameter task has been greatly automated by providing you with unlimited flexibility in specifying bunching requirements. Assigning the edge meshing parameters occurs after the development of the block topology model. This option is accessible by selecting **Meshing > Edge params**.

You can use the following pre-defined bunching laws or **Meshing laws**:

- Default (Bi-Geometric Law)
- Uniform
- Hyperbolic
- Poisson
- Curvature

- Geometric 1
- Geometric 2
- Exponential 1
- Exponential 2
- Bi-Exponential
- Linear
- Spline

You may modify these existing laws by applying pre-defined edge meshing functions, accessible through the **Meshing > Edge Params > Graphs** option in **Hexa**.

This option yields these possible functions:

- Constant
- Ramp
- S curve
- Parabola Middle
- Parabola Ends
- Exponential
- Gaussian
- Linear
- Spline

Note

By selecting the **Graphs** option, you may add/delete/modify the control points governing the function describing the edge parameter settings. Additional tools such as **Linked Bunching** and the multiple **Copy** buttons provide you with the ability to quickly apply the specified edge bunching parameters to the entire model.

Smoothing Techniques

In **Hexa**, both the block topology and the mesh may be smoothed to improve the overall block/mesh quality either in a certain region or for the entire model. The block topology may be smoothed to improve the block shape prior to mesh generation. This reduces the time required for development of the block topology model.

The geometry and its associative faces, edges, and points are all constraints when smoothing the block topology model. Once the block topology smoothing has been performed, you may smooth the mesh after specifying the proper edge bunching parameters.

The quality criteria for smoothing are described in the Help Manual, [under Blocking > Pre-Mesh Quality](#).

Refinement and Coarsening

The refinement function, which is found through **Blocking > Pre-Mesh Params > Refinement**, can be modified to achieve either a refined or a coarsened result. The refinement/coarsening may be applied in all three major directions simultaneously, or they may be applied in just one major direction.

Refinement

The refinement capability is used for solvers that accept non-conformal node matching at the block boundaries. The refinement capability is used to minimize the model size, while achieving proper mesh definition in critical areas of high gradients. Entering a scale factor greater than 1 will result in refinement.

Coarsening

In areas of the model where the flow characteristics are such that a coarser mesh definition is adequate, coarsening of the mesh may be appropriate to contain model size. Entering a scale factor less than 1 will result in coarsening.

Replay Functionality

Parametric changes made to model geometry are easily applied through the use of Hexa's replay functionality, found in **File > Replay Scripts**. Changes in length, width and height of specific geometry features are categorized as parametric changes. These changes do not, however, affect the block topology. Therefore, the **Replay** function is capable of automatically generating a topologically similar block model that can be used for the parametric changes in geometry.

If any of the Direct CAD Interfaces are used, all geometric parameter changes are performed in the native CAD system.

You can also use variables in the replay script to parametrize edge parameters. Refer to [Using Variables in the Replay Script](#) (p. 45) for details.

Generating a Replay File

The first step in generating a **Replay** file is to activate the recording of the commands needed to generate the initial block topology model. All of the steps in the mesh development process are recorded, including blocking, mesh size, edge meshing, boundary condition definition, and final mesh generation. The next step in the process is to make the parametric change in the geometry and then replay the recorded file on the changed geometry. All steps in the mesh generation process are automated from this point.

Advantage of the Replay Function

With the **Replay** option, you may analyze more geometry variations, thus obtaining more information on the critical design parameters. This can yield optimal design recommendations within the project time limits.

Using Variables in the Replay Script

You can use variables in the replay script as a means to parametrize edge parameters. An example of the use of variables in a replay script is as follows:

```
#variables
set n 10
set h1 0.01
set r1 1.2
```

```
ic_load_tetin myfile.tin
ic_hex_surface_blocking -inherited -swept -min_edge 0.0
ic_geo_new_family SOLID
ic_hex_twod_to_threed SOLID -swept
ic_hex_set_mesh 19 18 n $n h1 $h1 h2rel 0.0 r1 $r1 r2 2 lmax 0 default unlocked
ic_hex_create_mesh SURFS SOLID proj 2 dim_to_mesh 3
ic_hex_write_file hex.uns SURFS SOLID proj 2 dim_to_mesh 3 -family_boco family_boco.fbc
ic_uns_load hex.uns 3 0 {} 2
```

The variables for the edge parameters are set at the top of the replay file. Within the script, the '\$' indicates a variable. To parametrize the edge parameters, you may update the variables at the top of the script and then rerun the script.

Periodicity

Periodic definition may be applied to the model in **Hexa**. The **Periodic nodes** function, which is found under **Blocking > Periodic nodes**, plays a key role in properly analyzing rotating machinery applications, for example. Typically, you will model only a section of the rotating machinery, as well as implement symmetry, in order to minimize the model size. By specifying a periodic relationship between the inflow and outflow boundaries, the particular specification may be applied to the model -- flow characteristics entering a boundary must be identical to the flow characteristics leaving a boundary.

Applying the Periodic Relationship

The periodic relationship is applied to block faces and ensures that a node on the first boundary have two identical coordinates to the corresponding node on the second boundary. You will be prompted to select corresponding vertices on the two faces in sequence. When all vertices on both flow boundaries have been selected, a full periodic relationship between the boundaries has been generated.

Pre-Mesh Quality

The pre-mesh quality functions are accessible through **Blocking > Pre-Mesh Quality**. Applying any of the quality checks will yield a histogram plot. Some of the quality metrics are explained below.

Determining the Location of Elements

By clicking on any of the histogram bars with the left button, you may determine where in the model these elements are located. The selected histogram bars will be highlighted by a change in color. After selecting the bar(s), the **Show** button is pressed to highlight the elements in this range. If the **Solid** option is enabled, the elements marked in the histogram bars will be displayed with solid shading.

Determinant

The **Determinant** check computes the deformation of the elements in the mesh by first calculating of the Jacobian of each hexahedron and then normalizing the determinant of the matrix. A value of 1 represents a perfect hexahedral cube, while a value of 0 is a totally inverted cube with a negative volume. The mesh quality, measured on the x-axis, of all elements will be in the range from 0 to 1. If the determinant value of an element is 0, the cube has one or more degenerated edges. In general, determinant values above 0.3 are acceptable for most solvers.

The y-axis measures the number of elements that are represented in the histogram. This scale ranges from 0 to a value that is indicated by the **Height**. The subdivision among the quality range is determined by the number of assigned **Bars**.

Angle

The **Angle** option checks the maximum internal angle deviation from 90 degrees for each elements. Various solvers have different tolerance limits for the internal angle check. If the elements are distorted and the internal angles are small, the accuracy of the solution will decrease. It is always wise to check with the solver provider to obtain limits for the internal angle threshold.

Volume

The **Volume** check will compute the internal volume of the elements in the model. The units of the volume will be displayed in the unit that was used to create the model.

Warpage

The **Warpage** check will yield a histogram that indicates the level of element distortion. Nodes that are in-plane with one another will produce a element with small warpage. Nodes that make elements twisted or distorted will increase a element's distortion, giving a high degree of warpage. The y-axis is the scale for the number of elements represented in the histogram - a value determined by the assigned **Height**. The x- axis, which ranges from a **Min** of 0 to a **Max** of 90, is the degree of warpage that a elements experiences.

Properties

The **Properties** menu allows you to create different materials by specifying material or element properties, such as type, the Young's Modulus, and Poisson's ratio. Once the material is created, you can apply those properties to the elements.

Create Material Property

Here you can define a material by specifying a name of the material, define whether isotropic, enter in values for Young's Modulus, Shear modulus, Poisson's ratio, Mass Density, and Thermal expansion coefficient.

Save Material

This option allows you to save the material which is created so that you may retrieve the material whenever necessary. The material file will be saved with the .mat extension.

Open Material

This option allows you to open a material file so that you can use the same in future or modify the file and save for further usage.

Define Table

You can create different tables by specifying the values for x and y ,and can even visualize the graph.

Define Elements

Although the mesh has been created, you need to apply the material created to the respected elements. Different types of elements that can be defined include: Point, Line, Shell, and Volume. You have to specify the part, and various properties of the element.

Constraints

Here you can define the constraints on different entities such as points, curves, surfaces, or subsets, and define other options like **Contact definition**, **Velocity** and **Rigid Wall**.

Create Constraint / Displacement

This option allows you to apply a directional or rotational constraint on a an entity in any direction.

Define Contact

This option allows you to define contacts by Automatic Detection or Manual Definition.

Define Single Surface Contact

This is mainly used for LS-Dyna Solver, where you can pick the contact surface.

Define Initial Velocity

This allows you to define initial nodal point translational velocities by specifying the directional and rotational velocity for nodal sets.

Define Planar Rigid wall

You can define a Planar Rigid Wall by specifying the Head and Tail coordinates.

Loads

In this tab, there are several options available for applying internal and external loads, such as force, pressure, and temperature.

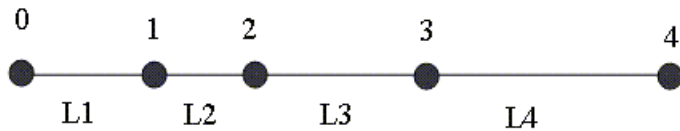
Theory

Force Distribution follows the following formulation.

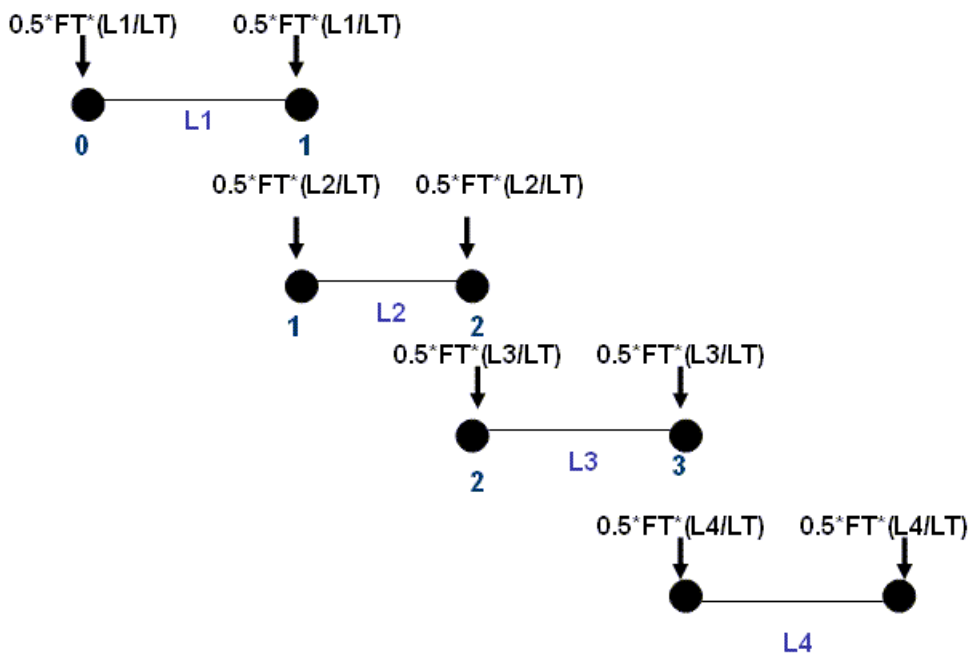
Curve

Total Force '**FT**' is applied on the curve as shown below, where L1 is the length of Element 1, '0' and '1' are the node number connecting Element 1.

Figure: Elements on Curve



Then the force distribution on 'Linear' elements as per the FEA concept is shown in the figure below.

Figure: Force Distribution as per the FEA concept

The formulation for a Linear Element is as follows.

Point 0: $F_0 = 0.5 \cdot FT \cdot (L1/LT)$

Point 1: $F_1 = 0.5 \cdot FT \cdot (L1/LT) + 0.5 \cdot FT \cdot (L2/LT)$

Point 2: $F_2 = 0.5 \cdot FT \cdot (L2/LT) + 0.5 \cdot FT \cdot (L3/LT)$

Point 3: $F_3 = 0.5 \cdot FT \cdot (L3/LT) + 0.5 \cdot FT \cdot (L4/LT)$

Point 4: $F_4 = 0.5 \cdot FT \cdot (L4/LT)$

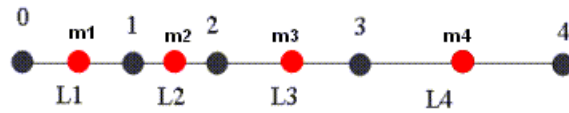
The general formula is:

$F_i = \text{Sum} [FT \cdot (L \text{ (attached element)}/LT) \cdot (1/\text{number of nodes per element})]$.

If you sum up **$F_0 + F_1 + F_2 + F_3 + F_4$** then the resultant comes to be **FT** . It also satisfies the FEA concepts.

The same force is applied on the **Quadratic Element** shown in [Figure : Quadratic Element Nodes position \(p. 55\)](#).

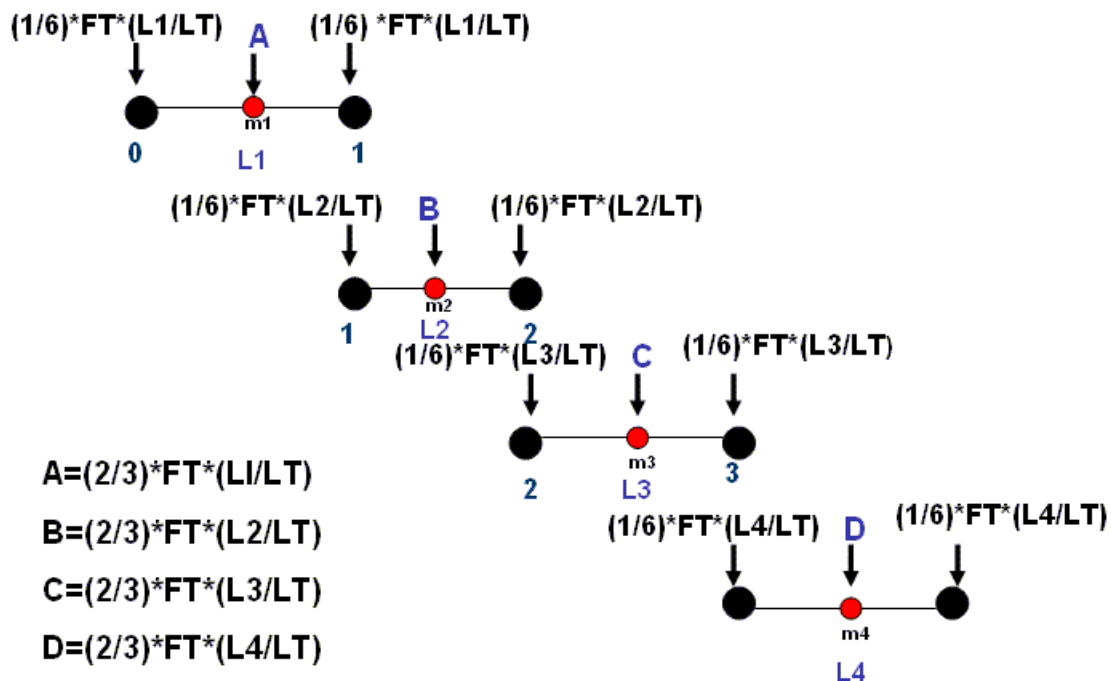
Figure: Quadratic Element Nodes position



m1,m2,m3,m4 are mid side nodes of Element 1,2,3,4

The Load distribution as per the FEA concept on the Quadratic elements is shown in [Figure : Load Distribution as per the FEA concepts](#) (p. 55).

Figure: Load Distribution as per the FEA concepts



The formulation of **Total Force** at the side nodes is as follows:

Point 0: $F0q = (1/6)* FT *(L1/LT)$

Point 1: $F1q = (1/6)* FT *(L1/LT) + (1/6)* FT *(L2/LT)$

Point 2: $F2q = (1/6)* FT *(L2/LT) + (1/6)* FT *(L3/LT)$

Point 3: $F3q = (1/6)* FT *(L3/LT) + (1/6)* FT *(L4/LT)$

Point 4: $F4q = (1/6)* FT *(L4/LT)$

But the formulation of mid-side node is:

Point m1: $\mathbf{Fm1} = (2/3) * \mathbf{FT} * (L1/LT)$

Point m2: $\mathbf{Fm2} = (2/3) * \mathbf{FT} * (L2/LT)$

Point m3: $\mathbf{Fm3} = (2/3) * \mathbf{FT} * (L3/LT)$

Point m4: $\mathbf{Fm4} = (2/3) * \mathbf{FT} * (L4/LT)$

As in the previous case of Linear Element the total Force of Quadratic element is:

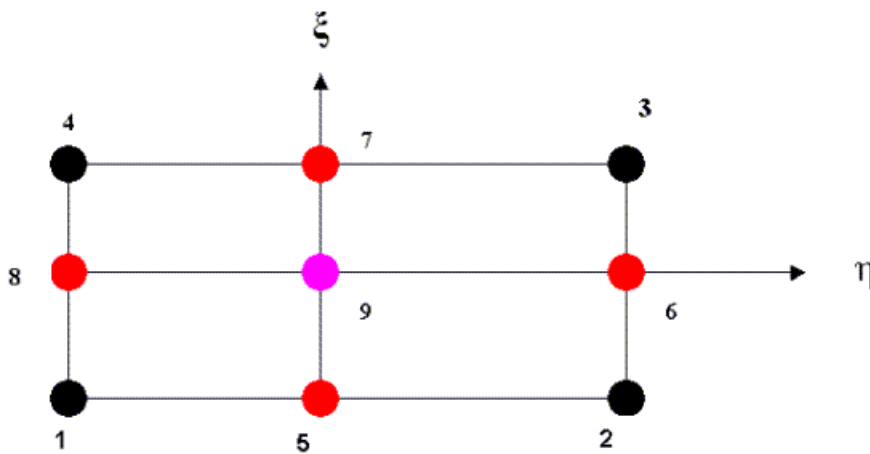
$$\mathbf{F_{Total}} = \mathbf{F0q} + \mathbf{F1q} + \mathbf{F2q} + \mathbf{F3q} + \mathbf{F4q} + \mathbf{Fm1} + \mathbf{Fm2} + \mathbf{Fm3} + \mathbf{Fm4},$$

which yields $\mathbf{F_{Total}} = \mathbf{FT}$.

QUAD 9 Consistent Nodal Load Distributions

The Nine Node two dimension Lagrange **QUAD 9** Element is shown in [Figure : QUAD9 Element \(p. 56\)](#).

Figure: QUAD9 Element



The Shape function for the **Corner Node**

$$\mathbf{N_1(\eta, \xi) = (1/4) * \eta(\eta-1)(\xi-1)}.$$

The Shape function for the **Mid-Side Node**

$$\mathbf{N_5(\eta, \xi) = (1/2) * \eta(1-\eta^2)(\xi-1)}.$$

The Shape function for the **Middle Node**

$$\mathbf{N_9(\eta, \xi) = (1-\eta^2)(1-\xi^2)}.$$

Suppose a Force \mathbf{F} is uniformly distributed over the whole Area. Then the pressure is

$$\mathbf{p = F/4}$$
 (Because in ' η, ξ ' coordinate system the **Area of the rectangle is '4'**).

To Find the Consistent Load:

Consistent Load at Node 1=L1

Consistent Load at Node 5=L5

Consistent Load at Node 9=L9

Upon integration we get the following values:

$$L1 = p \int_{-1}^1 \int_{-1}^1 N_1 d\xi d\eta = 4p/36$$

$$L5 = p \int_{-1}^1 \int_{-1}^1 N_5 d\xi d\eta = 4p/9$$

$$L9 = p \int_{-1}^1 \int_{-1}^1 N_9 d\xi d\eta = 16p/9$$

Now $F=4p$.

Putting this value in the above equation we get the consistent Nodal Force as

$$L1 = F/36$$

$$L5 = F/9$$

$$L9 = 4F/9$$

By symmetry **Consistent Nodal Force** on Node **1, 2, 3&4** are equal.

By symmetry **Consistent Nodal Force** on Node **5, 6, 7&8** are equal.

Now **Total Force** = $4*L1 + 4*L5 + L9$

$$= F/9 + 4F/9 + 4F/9$$

$$= F$$

Note

The same method is employed for calculating the Consistent Nodal Force on the **QUAD8** element, which gives correct results.

Force

Using this option, you can apply force or moments on entities in all three directions.

Forces can be applied by two different options. The Uniform option applies the force uniformly. For curves, this means that the nodes attached to the curve will have the same force applied to them. The Total option means that the force gets distributed among all the nodes of the selected entities as per FEA concepts.

Pressure

You can apply pressure loads to surfaces, subsets, or parts.

Temperature

This option allows you to apply temperature to points, curves, surfaces, bodies, and subsets.

Solve Options

Setup Solver Parameters has options for specifying the solver parameters. You can also specify the analysis solution and postprocessing parameters.

For more useful information about Solver Options, please go to [this page](#)..

Setup Solver Parameters

You can select from the following solvers: **ANSYS**, **Nastran**, **ABAQUS**, **AUTODYN**, and **LS-Dyna**.

Setup Analysis Type

Depending on the selected solver, different options are available. For the ANSYS solver, you can select either Structural or Thermal. If Nastran solver is selected, then you have the choice of more **Analysis** types.

Setup Sub-Case

You can create subcases to apply the load in different steps.

Write/View Input file

You can create and view the input file generated for the solver.

Submit Solver Run

Using this option, you can solve the input file generated for a particular solver.

Post Process Results

This option allows you to plot the results.

FEA Solver Support

For more useful information on FEA Solvers Support, please go to [this page](#).

Cart3D

For FAQ on Cart3D, including information on Cart3D batch commands, and sixDOF batch commands, please go to [this page](#).

For other product information, please go to [this page](#).

Workbench Integration

Named Selections

Named Selections created in Design Modeler can be imported into Advanced Meshing as either parts or subsets. In Design Modeler, make sure that the Named Selections option under the Default Geometry Options is checked. To import all Named Selections, leave the prefix field blank. Then proceed with opening the project in Advanced Meshing. In Advanced Meshing, go to Settings > Workbench. Select the **Named Selections as parts** option, or toggle it OFF to import the named selections as subsets.

